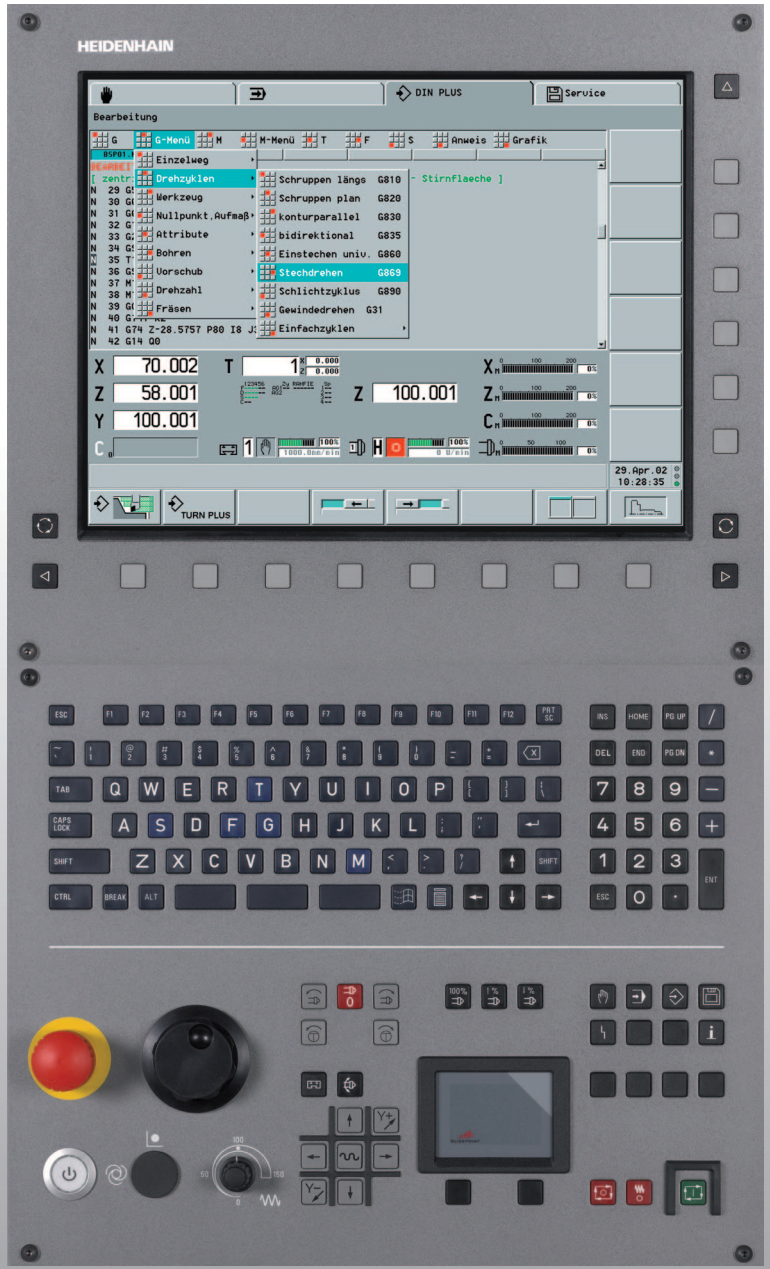
HEIDENHAIN

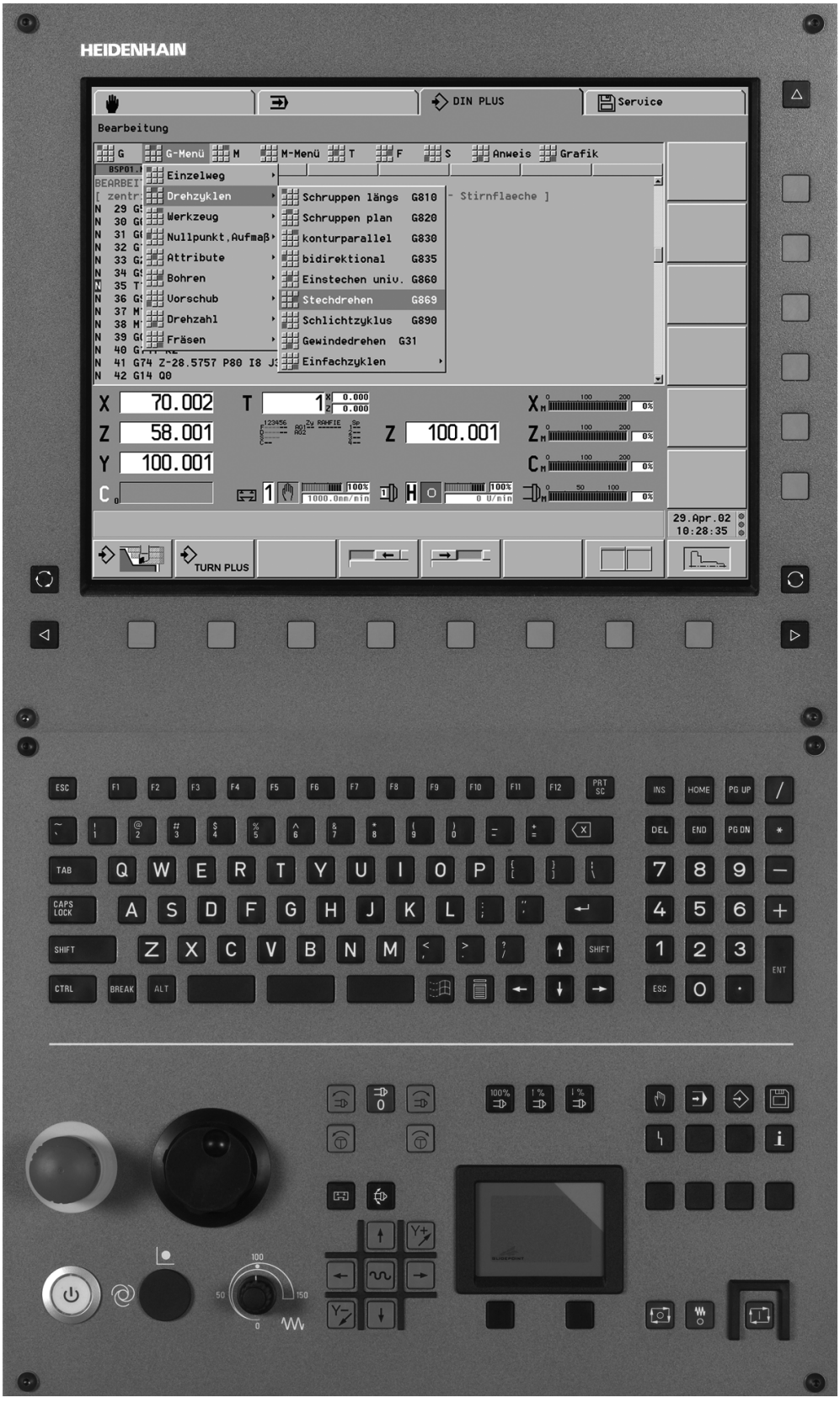


User's Manual

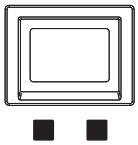
CNC Pilot 4290

NC Software
625 952-xx
V7.1

English (en)
4/2010


HEIDENHAIN
DIN PLUS
Service
Bearbeitung
G
G-Menü
M
M-Menü
T
F
S
Anweis
Grafik
Einzelweg
Drehzyklen
Werkzeug
Nullpunkt,Aufmaß
Attribute
Bohren
Vorschub
Drehzahl
Fräsen
Schruppen längs G810
Schruppen plan G820
konturparallel G830
bidirektional G835
Einstechen univ. G860
Stechdrehen G869
Schlichtzyklus G890
Gewindedrehen G31
Einfachzyklen
- Stirnflaeche]
N 41 G74 Z-28.5757 P80 I8
N 42 G14 Q0
X 70.002
Z 58.001
Y 100.001
C
T 1
Z 100.001
29.Apr.02
10:28:35
TURN PLUS

Data input keypad	Machine operating panel
Manual control operating mode	Cycle start
Automatic operating mode	Cycle stop
Programming modes (DIN PLUS, simulation, TURN PLUS)	Feed rate stop
Organization modes (parameter, service, transfer)	Spindle STOP
Display error status	Spindle on - M3/M4 direction
Call the info system	Spindle jog - M3/M4 direction (The spindle turns until you press the key.)
ESC ■ Back by one menu level ■ Close dialog box, do not save data	Manual direction keys +X/–X
INS ■ Insert list element ■ Close dialog box, save data	Manual direction keys +Z/–Z
ALT ■ Edits a list element	Manual direction keys +Y/–Y
DEL ■ Deletes the list element ■ Deletes the selected character or the character to the left of the cursor	Rapid-traverse key
0 . . . 9 Numerals for entering values and selecting soft keys	Slide change key
Decimal point	Spindle change key
Minus as algebraic sign	Spindle speed at the programmed value
"Continue key" for special functions (e.g. marking)	Increase/decrease spindle speed by 5%
Cursor keys	Override button for feed-rate override

Data input keypad	Machine operating panel
PgUp PgDn Page up, Page down (PgUp/PgDn) ■ Change to previous/next screen page ■ Change to previous/next screen dialog box ■ Switches between input windows	 Touchpad with right and left mouse key
Enter Enter – Confirmation of input	

CNC PILOT 4290, Software and Functions

This manual describes functions that are available in the CNC PILOT 4290 with NC software number 625 (Release 7.1). For programming the B and Y axes, please refer to the User's Manual "CNC PILOT 4290 with B and Y Axes". It is not described in this manual.

The machine manufacturer adapts the features offered by the control to the capabilities of the specific lathe by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by the CNC PILOT on your machine tool.

CNC PILOT functions that may not be available on your machine include:

- Machining with the C Axis
- Machining with the B axis
- Machining with the Y-axis
- Full-surface machining
- Tool monitoring
- Graphically supported interactive contour definition
- Automatic or graphically supported interactive DIN PLUS program generation

Please contact your machine manufacturer for detailed information on the features that are supported by your machine tool.

Many machine manufacturers and HEIDENHAIN offer programming courses for the CNC PILOT controls. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other CNC PILOT users.

HEIDENHAIN also offers the PC software DataPilot 4290, which is designed for use with the CNC PILOT 4290. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes.

Intended place of operation
The CNC PILOT 4290 complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Contents

Introduction and Fundamentals 1
Basics of Operation 2
Manual Control and Automatic Modes 3
ISO Programming 4
Graphic Simulation 5
TURN PLUS 6
Parameters 7
Operating Resources 8
Service and Diagnosis 9
Transfer 10
Tables and Overviews 11

1 Introduction and Fundamentals 29

1.1 The CNC PILOT 30
Programming 30
The C axis 31
The Y axis 32
Full-surface machining 33
The B axis 34
1.2 The Modes of Operation 35
1.3 Expansion Stages (Options) 37
1.4 Fundamentals 39
Position encoders and reference marks 39
Axis designations and coordinate system 40
Machine reference points 40
Absolute and incremental workpiece positions 41
Units of measure 42
1.5 Tool Dimensions 43

2 Basics of Operation 45

2.1 User Interface 46
Screen displays 46
Controls and displays 47
Selecting the operating mode 48
Data input, selection of functions 48
2.2 Info and Error System 50
The info system 50
Context-sensitive help 52
Direct error messages 52
Error display 53
Additional information on error messages 54
PLC display 54
2.3 Data Backup 55
2.4 Explanation of Terms 56

3 Manual Control and Automatic Modes 57

3.1 Switch-On, Switch-Off, Reference Run 58
Switch-on 58
Reference run for all axes 58
Reference jog for single axis 59
Monitoring EnDat encoders 59
Switch-off 60

3.2 Manual Control Mode 61
Entering the machine data 62
M commands in Manual Control mode 63
Manual turning operations 64
Handwheel 65
Spindle and manual direction keys 65
Slide/spindle change key 66
3.3 Table for Tools and Chucking Equipment 67
Setting up a tool list 68
Comparing a tool list with an NC program 70
Transferring the tool list from an NC program 71
Simple tools 71
Tool life management 72
Setting up the chucking table 74
3.4 Setup Functions 75
Setting the tool changing point 75
Shifting the workpiece zero point 76
Defining the protection zone 77
Setting up machine dimensions 78
Tool measurement 79
Calculate the tool compensation 80
3.5 Automatic Mode 81
Program selection 82
Finding a start block 84
Modifying the program run 85
Compensation 87
Tool life management 88
Inspection mode 89
Block display, variable output 93
Graphic display 94
Mechatronic tailstock 95
Post-process measuring status 96
3.6 Machine Display 97
Switching the display 97
Display elements 97
3.7 Load Monitoring 100
Machining using load monitoring 101
Reference machining 102
Production using load monitoring 103
Editing limit values 103
Analyzing reference machining 104
Load monitoring parameters 105

4 DIN Programming 107

4.1 DIN Programming 108
Introduction 108
DIN PLUS screen 109
Linear and rotary axes 110
Units of measurement 111
Elements of a DIN program 111
4.2 Programming Notes 113
Configuring the DIN editor 113
Parallel editing 114
Selecting submenus, positioning the cursor 114
Making, editing and deleting NC blocks 115
Search functions 116
Conversational or free editing 117
Geometry and machining commands 117
Contour programming 118
List of G functions 120
Address parameters 120
Tool programming 121
Subprograms, expert programs 122
NC program conversion 122
Fixed cycles 123
4.3 The DIN PLUS Editor 124
Overview of main menu 124
Overview of geometry menu 125
Overview of machining menu 126
New NC program 127
NC program management 128
Graphics window 129
Workpiece-blank programming 130
Block numbering 130
Programming instructions 131
Block group menu 133

4.4 Program Section Code 135
PROGRAM HEAD section 136
TURRET section 137
CHUCKING EQUIPMENT section 142
CONTOUR section 143
BLANK section 143
FINISHED PART section 143
AUXILIARY CONTOUR section 144
FRONT section 144
REAR SIDE section 144
SURFACE section 144
MACHINING section 144
END code 144
ZUORDNUNG [ASSIGNMENT] instruction $.. 144
SUBPROGRAM section 145
RETURN code 145
CONST code 145
4.5 Definition of Workpiece Blank 146
Chuck piece: bar/tube G20-Geo 146
Casting G21-Geo 146
4.6 Basic Contour Elements 147
Starting point of turning contour G0-Geo 147
Line segment in a contour G1-Geo 147
Circular arc of turning contour G2/G3-Geo 148
Circular arc of turning contour G12/G13-Geo 150
4.7 Contour Form Elements 152
Recess (standard) G22-Geo 152
Recess (general) G23-Geo 153
Thread with undercut G24-Geo 155
Undercut contour G25-Geo 156
Thread (standard) G34-Geo 159
Thread (general) G37-Geo 160
Hole (centric) G49-Geo 162
4.8 Attributes for Contour Description 163
Precision stop 164
Surface roughness G10-Geo 164
Feed rate reduction factor G38-Geo 165
Attributes for superimposed elements G39-Geo 165
Blockwise oversize G52-Geo 166
Feed per revolution G95-Geo 166
Additive compensation G149-Geo 167
4.9 C-Axis Contours—Fundamentals 168
Milling contour position 168
Circular pattern with circular slots 169

4.10 Front and Rear Face Contours 172
Starting point of front/rear face contour G100-Geo 172
Line segment in front/rear face contour G101-Geo 172
Circular arc in front/rear face contour G102/G103-Geo 173
Bore hole on front/rear face G300-Geo 174
Linear slot on front/rear face G301-Geo 175
Circular slot on front/rear face G302/G303-Geo 175
Full circle on front/rear face G304-Geo 176
Rectangle on front/rear face G305-Geo 176
Eccentric polygon on front/rear face G307-Geo 177
Linear pattern on front/rear face G401-Geo 177
Circular pattern on front/rear face G402-Geo 178
4.11 Lateral Surface Contours 179
Starting point of lateral surface contour G110-Geo 179
Line segment in a lateral surface contour G111-Geo 179
Circular arc in lateral surface contour G112-/G113-Geo 180
Hole on lateral surface G310-Geo 181
Linear slot on lateral surface G311-Geo 182
Circular slot on lateral surface G312/G313-Geo 182
Full circle on lateral surface G314-Geo 183
Rectangle on lateral surface G315-Geo 183
Eccentric polygon on lateral surface G317-Geo 184
Linear pattern on lateral surface G411-Geo 185
Circular pattern on lateral surface G412-Geo 186
4.12 Tool Positioning 187
Rapid traverse G0 187
Setting the tool change position G14 187
Rapid traverse to machine coordinates G701 188
4.13 Simple Linear and Circular Movements 189
Linear path G1 189
Circular path G2/ G3 190
Circular path G12/ G13 191
4.14 Feed Rate and Spindle Speed 192
Rotational speed limiting G26 192
Acceleration (slope) G48 192
Interrupted feed G64 193
Feed per minute for rotary axes G192 193
Feed per tooth Gx93 194
Constant feed rate G94 (feed per minute) 194
Feed per revolution Gx95 194
Constant surface speed Gx96 195
Speed Gx97 195

4.15 Tool-Tip and Cutter Radius Compensation 196
G40: Switch off TRC/MCRC 197
G41/G42: Switch on TRC/MCRC 197
4.16 Zero Point Shifts 198
Zero point shift G51 199
Parameter-dependent zero offset G53, G54, G55 199
Additive zero point shift G56 200
Absolute zero point shift G59 201
Mirror/shift contour G121 202
4.17 Oversizes 204
Switch off oversize G50 204
Axis-parallel oversize G57 204
Contour-parallel oversize (equidistant) G58 205
4.18 Safety Clearances 206
Safety clearance G47 206
Safety clearance G147 206
4.19 Tools, Types of Compensation 207
Tool call – T 207
(Changing the) tool edge compensation G148 208
Additive compensation G149 209
Compensation of right-hand tool tip G150
Compensation of left-hand tool tip G151 210
Adding tool dimensions G710 211
4.20 Contour-Based Turning Cycles 212
Working with cycles 212
Longitudinal roughing G810 212
Face roughing G820 215
Contour-parallel roughing G830 218
Contour-parallel with neutral tool G835 220
Recessing G860 222
Recessing cycle G866 224
Recess turning cycle G869 225
Finish contour G890 228
4.21 Simple Turning Cycles 231
End of cycle G80 231
Simple longitudinal roughing G81 231
Simple face roughing G82 232
Simple contour repeat cycle G83 234
Undercut cycle G85 235
Recessing G86 236
Radius cycle G87 238
Chamfer cycle G88 238

4.22 Thread Cycles 239
Thread switch G933 239
Thread cycle G31 240
Simple thread cycle G32 242
Thread single path G33 244
4.23 Drilling Cycles 246
Drilling cycle G71 246
Boring, countersinking G72 248
Tapping G73 249
Tapping G36 250
Deep-hole drilling G74 251
4.24 C-Axis Commands 253
No. of C axis G119 253
Reference diameter G120 253
Zero point shift, C axis G152 254
Standardize C axis G153 254
4.25 Front/Rear-Face Machining 255
Rapid traverse on front/rear face G100 255
Linear segment on front/rear face G101 256
Circular arc on front/rear face G102/G103 257
4.26 Lateral Surface Machining 258
Rapid traverse, lateral surface G110 258
Line segment on lateral surface G111 259
Circular arc on lateral surface G112/G113 260
4.27 Milling Cycles 261
Contour milling G840—Fundamentals 261
Pocket milling, roughing G845 – Fundamentals 270
Pocket milling, finishing G846 276
Thread milling, axial G799 278
Engraving on front face G801 279
Engraving on lateral surface G802 280
Character set for engraving 280

4.28 Assignment, Synchronization, Workpiece Transfer 282
Converting and mirroring G30 282
Spindle with workpiece G98 283
Workpiece group G99 284
One-sided synchronization G62 284
Synchronization marking G162 285
Synchronous start of slides G63 285
M97 Synchronous function 286
Spindle synchronization G720 286
C-angle offset G905 287
Measuring angular offset during spindle synchronization G906 288
Traversing to a fixed stop G916 288
Controlled parting using lag error monitoring G917 291
Controlled parting using spindle monitoring G991 292
Values for controlled parting G992 293
4.29 Contour Follow-Up 294
Saving/loading contour follow-up G702 294
Contour follow-up G703 294
K default branch G706 295
4.30 In-process and Post-process Measuring 296
In-process measuring 296
Post-process measurement G915 298
4.31 Load Monitoring 300
Fundamentals of load monitoring 300
Specifying the monitoring zone G995 301
Type of monitoring G996 301

4.32 Other G Functions 302
Period of dwell G4 302
Precision stop G7 302
Precision stop off G8 302
Precision stop G9 302
Move rotary axis G15 303
Switch off protection zone G60 303
Chucking equipment in simulation G65 304
Component position G66 305
Waiting for time G204 305
Update nominal values G717 305
Move lag error G718 306
Actual values in variables G901 306
Zero-point shift in variables G902 306
Lag error in variables G903 306
Block speed monitoring off G907 306
Feed rate override 100% G908 307
Interpreter stop G909 307
Velocity feedforward G918 307
Spindle override 100% G919 307
Deactivate zero-point shifts G920 308
Deactivate zero-point shifts, tool lengths G921 308
T no. internal G940 308
Transferring magazine compensation values G941 309
Servo lag limit G975 309
Activating zero-point shifts G980 309
Activate zero-point shifts, tool lengths G981 310
Sleeve monitoring G930 310
Shaft speed with V constant G922 311
4.33 Data Input and Data Output 312
Output window for # variables WINDOW 312
Input of # variables INPUT 312
Output of # variables PRINT 313
V variable simulation 313
Output window for V variables WINDOWA 313
Input of V variables INPUTA 314
Output of V variables PRINTA 314
4.34 Programming with Variables 315
variables 316
V variables 318

4.35 Conditional Block Run 322
Program branching IF..THEN..ELSE..ENDIF 322
WHILE..ENDWHILE program repeat 323
SWITCH..CASE—program branching 324
Skip level /.. 326
Slide code $.. 326
4.36 Subprograms 327
Subprogram call: L"xx" V1 327
Dialog texts in subprogram call 328
Help graphics for subprogram calls 329
4.37 M Commands 330
M commands for program-run control 330
Machine commands 331
4.38 Lathes with Multiple Slides 332
Multi-slide programming 332
Program run 334
Positioning the steady rest 334
Traveling steady rest 336
Two slides work simultaneously 338
Two slides work in succession 340
Machining with four-axis cycle 342
4.39 Full-surface machining 344
Fundamentals of full-surface machining 344
Programming of full-surface machining 345
Full-surface machining with opposing spindle 346
Full-surface machining with single spindle 349
4.40 DIN PLUS Program Example 351
Example of a subprogram with contour repetitions 351
4.41 DIN PLUS Templates 354
Starting templates 354
Structure templates 354
Design of structure templates 355
Transfer parameters for structure templates 355
Editing structure templates 356
Help graphics for structure templates 356
Template menu 356
Template example 357
4.42 Connection between Geometry and Machining Commands 359
Turning 359
C-axis machining – front/rear face 360
C-axis machining – lateral surface 360

5 Graphic Simulation 361

5.1 Simulation Mode of Operation 362
- Screen layout, soft keys 363
- Graphic elements 364
- Displays 364
- Zero point shifts 366
- Path display 367
- Simulation window 368
- Setting the simulation window 369
- Configuring the simulation 370
- Adjusting the section (zoom function) 371
- Errors and warnings 372
- How to activate the simulation function 372
- Simulation mode 373

5.2 Contour Simulation 374
- Functions of the contour simulation 374
- Contour dimensioning 375

5.3 Machining Simulation 376
- Checking the workpiece machining 376
- Protection zone and limit switch monitoring (machining simulation) 377
- Dynamic limit switch monitoring 377
- Contour checking 378
- Saving the generated contour 378
- Displaying the tool tip reference point 379

5.4 Motion Simulation 380
- Real-time simulation 380
- Protection zone and limit switch monitoring (motion simulation) 381
- Contour checking 381

5.5 3-D View 382
- Influencing the 3-D view 382

5.6 Debugging Functions 383
- Simulation with starting block 383
- Displaying variables 384
- Editing variables 385

5.7 Checking Multi-channel Programs 386

5.8 Time Calculation, Synchronous Point Analysis 387
- Time Calculation 387
- Synchronous point analysis 388

6 TURN PLUS 389

6.1 TURN PLUS Mode of Operation 390
TURN PLUS concept 390
TURN PLUS files 391
TURN PLUS program management 391
Operating notes 392
6.2 Program Head 393
Generating programs for automatic lathes 394
6.3 Workpiece Description 396
Entering the workpiece blank contour 396
Entering the finished part contour 397
Superimposing form elements 398
Integrating overlay elements 399
Entering contours machined with the C axis 400
6.4 Contours of Workpiece Blanks 402
Bar 402
Tube 402
Cast blank (or forged blank) 403
6.5 Contours of Finished Parts 404
Notes on defining contours 404
Starting point of contour 404
Linear elements 405
Circular element 406
6.6 Form Elements 408
Chamfer 408
Rounding 408
Undercut type E 409
Undercut type F 409
Undercut type G 409
Undercut type H 410
Undercut type K 410
Undercut type U 410
Recess general 411
Recess type D (sealing ring) 412
Relief turn (type FD) 413
Recess type S (guarding ring) 413
Thread 414
(Centric) Hole 415
6.7 Overlay Elements 418
Circular arc 418
Wedge/rounded wedge 418
Pontoon 419
Linear superimposition (“linear overlay”) 419
Circular superimposition (“circular overlay”) 420

6.8 C-Axis Contours 421
Position of a front or rear face contour 421
Position of a lateral surface contour 421
Milling depth 421
Entering the C-axis contour dimensions 422
Front or rear face: Starting point 422
Front or rear face: Linear element 423
Front or rear face: Circular element 424
Front or rear face: Single hole 426
Front or rear face: Circle (full circle) 428
Front or rear face: Rectangle 429
Front or rear face: Polygon 430
Front or rear face: Linear slot 431
Front or rear face: Circular slot 432
Front or rear face: Linear hole or figure pattern 433
Front or rear face: Circular hole or figure pattern 434
Lateral surface: Starting point 435
Lateral surface: Linear element 436
Lateral surface: Circular element 437
Lateral surface: Single hole 438
Lateral surface: Circle (full circle) 440
Lateral surface: Rectangle 441
Lateral surface: Polygon 442
Lateral surface: Linear slot 443
Lateral surface: Circular slot 444
Lateral surface: Linear hole or figure pattern 445
Lateral surface: Circular hole or figure pattern 446
6.9 Help Functions 447
Unresolved contour elements 447
Selections 448
Zero point shift 452
Copying a contour section in linear series 452
Copying a contour section in circular series 453
Copying a contour section by mirroring 453
Calculator 454
Digitizing 455
Checking contour elements (inspector) 456
Error messages 457
6.10 Importing of DXF Contours 458
Fundamentals for DXF import 458
Configuring the DXF Import 459
DXF import 460

6.11 Manipulating Contours 461
Editing the contours of a blank part 461
Deleting contour elements 462
Editing contour elements or form elements 462
Adding a contour or contour element 463
Closing the contour 464
Resolving a contour 464
Trimming – Linear element 465
Trimming – Length of contour 466
Trimming – Radius of arc 466
Trimming – Diameter of linear element 467
Transformations – Fundamentals 467
Transformations – Shifting 468
Transformations – Rotating 468
Transformations – Mirroring 469
Transformations – Inverting 469
6.12 Assigning Attributes 470
Attributes for workpiece blanks 470
Attributes – Oversize 471
Attributes – Feed rate 472
Attributes – Peak-to-valley 472
Attributes – Additive compensation 473
Machining attributes – Measure 473
Machining attributes – Threading 474
Machining attributes – Drill – Retraction plane 475
Machining attributes – Drilling combinations 475
Machining attributes – Contour milling 476
Machining attributes – Area milling 477
Machining attributes – Deburring 478
Machining attributes – Engraving 479
Machining attributes – Precision stop 479
Machining attributes – Separation point 480
Attributes – Exclusion from machining 480
Deleting machining attributes 481

6.13 Preparing a Machining Process 482
Preparing a machining process – Fundamentals 482
Chucking a workpiece at the spindle 483
Chucking a workpiece at the tailstock 483
Defining the cutting limit 484
Deleting the chucking data 484
Rechuck – Standard machining 485
Rechuck – 1st setup after 2nd setup 486
Parameters for two-jaw, three-jaw or four-jaw chucks 488
Collet chuck parameters 489
Parameters for face drivers (“without chuck”) 489
Parameters for face drivers with jaws (“Three-jaw chuck indirect”) 490
Setting up and managing tool lists 490

6.14 Interactive Working Plan Generation (IWG) 494
Working plan exists 495
Generating a work block 496
Calling a tool 497
Cutting data 497
Cycle specification 498
Overview of roughing operations 499
Roughing longitudinal (G810) 500
Roughing transverse (G820) 501
Roughing contour-parallel (G830) 502
Residual roughing – longitudinal 503
Residual roughing – transverse 504
Residual roughing – contour-parallel 505
Roughing hollowing – neutral tool (G835) 506
Overview of recessing operations 507
Contour recessing radial/axial (G860) 508
Recessing radial/axial (G866) 509
Recess turning radial/axial (G869) 510
Parting 512
Parting and workpiece transfer 513
Overview of drilling operations 516
Centric predrilling (G74) 517
Centering, countersinking (G72) 518
Drilling, reaming, deep-hole drilling 519
Tapping 520
Finishing 521
Finishing – Clearance turning 524
Finishing – undercut 524
Thread machining (G31) 525
Overview of milling operations 526
Contour milling – Roughing/Finishing (G840) 527
Deburring (G840) 529
Engraving (G840) 530
Pocket milling – Roughing/Finishing (G845/G846) 531
Special machining (SM) 532
6.15 Automatic Working Plan Generation (AWG) 534
Generating a working plan 534
Machining sequence – Fundamentals 535
Editing and managing machining sequences 536
Overview of machining sequences 538
6.16 Control Graphics 547
Adjusting the section (zoom function) 547
Setting the control graphics 548

6.17 Configuring TURN PLUS 549
General settings 549
Configuring windows (views) 550
Configuring the control graphics 550
Setting the coordinate system 551
6.18 Machining Information 552
Tool selection, turret assignment 552
Contour recessing, recess turning 553
Drilling 553
Cutting data, coolant 553
Hollowing 554
Inside contours 555
Drilling 556
Shaft machining 557
Multi-slide machines 559
Full-surface machining 560
6.19 Example 562
Creating a program 562
Defining the workpiece blank 563
Defining the basic contour 563
Defining form elements 564
Preparing the machining process, chucking 565
Generating and saving a working plan 565

7 Parameters 567

7.1 The Parameter Mode of Operation 568
7.2 Editing Parameters 569
Current parameters 569
Parameter lists 569
Editing configuration parameters 570
7.3 Machine Parameters (MP) 571
General machine parameters 571
Machine parameters for slides 572
Machine parameters for spindles 573
Machine parameters for C axes 574
Machine parameters for linear axes 575
7.4 Control Parameters 577
General control parameters 577
Control parameters for simulation 579
Control parameters for machine display 580
7.5 Set-Up Parameters 583

7.6 Machining Parameters 585
1 – Global parameters for finished parts 585
2 – Global technology parameters 586
3 – Centric predrilling 588
4 – Roughing 591
5 – Finishing 594
6 – Recessing and contour recessing 597
7 – Thread cutting 599
8 – Measuring 600
9 – Drilling 600
10 – Milling 602
Load monitoring 603
20 – Direction of rotation for rear-side machining 604
21 – Name of the subroutines 605
22 – Sequence of tool selection 605
23 – Template management 606
24 – Parameter of the rechucking subroutines 606

8 Operating Resources 607

8.1 Tool Database 608
Tool editor 608
Overview of tool types 612
Tool parameters 614
Tool holder, tool mount 624
8.2 Database for Chucking Equipment 628
Chucking equipment editor 628
Chucking equipment lists 629
Chucking equipment data 630
8.3 Technology Database 641
Editing the technology data 642
Cutting-value tables 643

9 Service and Diagnosis 645

9.1 The Service Mode of Operation 646
9.2 Service Functions 647
Access authorization 647
System service 648
Fixed-word lists 649
9.3 Maintenance System 650
Maintenance dates and intervals 651
Displaying maintenance actions 652
9.4 Diagnosis 655
Information and display 655
Log files and network settings 656
Software update 657

10 Transfer 659

10.1 The Transfer Mode of Operation 660
Overview of data transfer methods 661
Configuring Windows networks 663
Configuring the serial interface or “printer” 666
10.2 Data Transfer 668
Enabling, file types 668
Basics of operation 669
Transmitting and receiving files 671
10.3 Parameters and Operating Resources 674
Transmitting parameters/operating resources 675
Loading parameters/operating resources 676
Backing up and restoring data 677
Viewing parameter, operating-resource or backup files 679
10.4 File Organization 680
Fundamentals for file organization 680
Managing files 681

11 Tables and Overviews 683

11.1 Undercut and Thread Parameters 684
Undercut DIN 76, Parameters 684
Undercut DIN 509 E, Parameters 686
Undercut DIN 509 F, Parameters 686
Thread Parameters 687
Thread pitch 688
11.2 Pin Layouts and Connecting Cables for the Data Interfaces 694
RS-232-C/V.24 interface for HEIDENHAIN devices 694
Non-HEIDENHAIN devices 695
RS-422/V.11 interface 696
Ethernet interface RJ45 socket 696
11.3 Technical Information 697
Specifications 697
Accessories 698
User functions 698



1

Introduction and Fundamentals

1.1 The CNC PILOT

The CNC PILOT is a contouring control designed for complex lathes and turning centers. In addition to turning operations, the control can also perform milling, drilling and boring operations. The C, Y and B axes enable you to drill and mill on the front and rear faces, the lateral surface and oblique planes. And as always, the CNC PILOT supports full-surface machining with dual-spindles.

The CNC PILOT controls up to six slides, four spindles, two C axes, one B axis and a pocket-oriented tool magazine. The control can machine up to four workpieces simultaneously.



Programming

Depending on the type and complexity of the parts to be machined and your organization, you can choose the type of programming best suited to your tasks.

In **TURN PLUS** you describe the contour of the blank and finished part using interactive graphics. Then you call the automatic working plan generation (AWG), and the NC program will be generated fully automatically on a keystroke. An alternative is the interactive working plan generation (IWG). When using the IWG, you determine the sequence of machining and other technical details.

Every working step is shown in the control graphics and can be corrected immediately. The result of program creation with TURN PLUS is a structured DIN PLUS program.

TURN PLUS minimizes the number of entries required, but it requires that the the tool data and cutting data has already been entered.

If TURN PLUS fails to create the optimal NC program for technologically sophisticated machining operations, or if you primarily want to reduce the machining time, write the NC program with DIN PLUS.

In **DIN PLUS** you first describe the contour of the workpiece blank and finished part. The “simplified geometry programming” calculates nondimensioned coordinates if, for example, the drawing is not dimensioned for conventional NC. Then you write the NC program using powerful fixed cycles.

Both TURN PLUS and DIN PLUS support a C or Y axis and full-surface machining. DIN PLUS cycles are available for working with the B axis.

As an alternative, you can machine your workpiece in DIN PLUS with linear and circular movements and simple turning cycles, as you are accustomed to in conventional DIN programming.

The **graphic simulation** feature enables you to subject your NC programs to a realistic test. The CNC PILOT displays the machining of up to four workpieces in the working space. The simulation shows workpiece blanks and finished parts, chucking equipment and tools to scale. When working with the tilted B axis, the working plane is also shown tilted. This enables you to see, without distortion, the holes and milling contours to be machined.

You can program your NC programs and test them—even during machining operations—directly on the machine.

Regardless of whether you are machining a simple or complex part, producing a single part or a series of parts, or a whole batch on a turning center, the CNC PILOT always gives you optimum support.

The C axis

With a C axis you can drill and mill a workpiece on its front, back and lateral surfaces.

When the C axis is used, one axis interpolates linearly or circularly with the spindle in the given working plane, while the third axis interpolates linearly.

The CNC PILOT supports part program creation with the C axis in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation



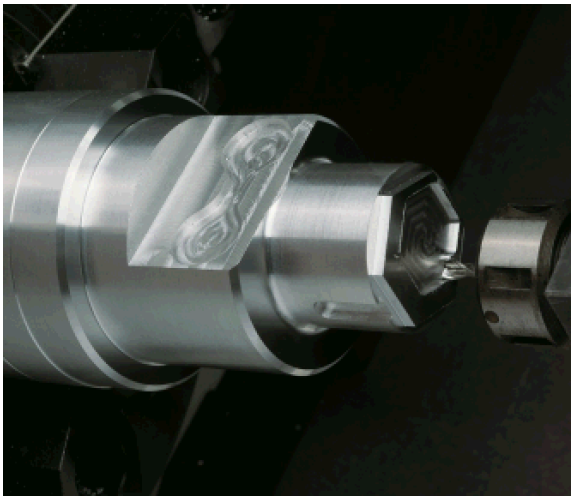
The Y axis

With a Y axis you can drill and mill a workpiece on its front, back and lateral surfaces.

During use of the Y-axis, two axes interpolate linearly or circularly in the given working plane, while the third axis interpolates linearly. This enables you to machine slots or pockets, for example, with plane floors and perpendicular edges. By defining the spindle angle, you can determine the position of the milling contour on the workpiece.

The CNC PILOT supports part program creation with the Y axis in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation



Full-surface machining

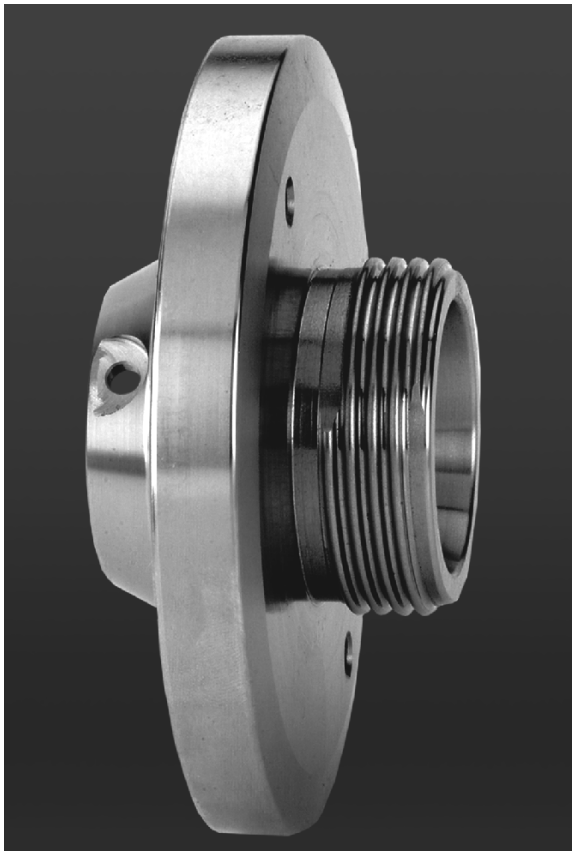
Functions like angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation ensures efficient machining as well as simple programming of full-surface machining.

The functions for full-surface machining are available in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation

The CNC PILOT supports full-surface machining for all common machine designs.
Examples: Lathes with

- Rotating gripper
- Movable opposing spindle
- Several spindles, slides and tool carriers



The B axis

The **B axis** makes it possible to drill, bore and mill in oblique planes. To make programming easy, the coordinate system is tilted in such a way that you can define the drilling patterns and milling contours in the YZ plane. The actual drilling or milling operation is then performed in the tilted plane.

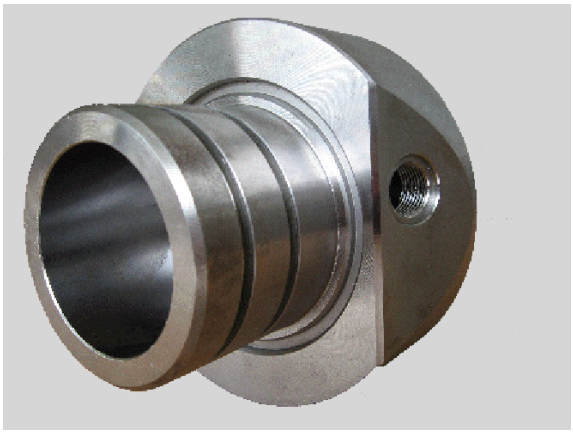
During work on the tilted plane, the tool is perpendicular to the plane. The tilting angle of the B axis and the angle of the tilted plane are identical.

Another advantage of the B axis is the flexible use of tools during turning operations. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and opposing spindles.

In this way, you need fewer tools and fewer tool changes.

The CNC PILOT supports part program creation with the B axis in DIN PLUS.

The **graphical simulation** shows the machining operation in a tilted working plane in the familiar lathe and front windows, as well as in the "side view (YZ)."



User's Manual for the B and Y axes

The manual and automatic control functions a well as the programming and testing of NC programs for the B and Y axes are described in a separate User's Manual. Please contact HEIDENHAIN if you require a copy of this User's Manual.

1.2 The Modes of Operation

Operating modes	
	**Manual mode:** In the Manual Control mode you set up the machine and move the axes manually.
	**Automatic mode:** NC in Automatic mode, NC programs are executed from start to end. You control and monitor the machining of the workpieces.
	**DIN PLUS programming mode:** You write the structured NC programs in DIN PLUS. First you define the geometry of the blank and finished part, and then program the machining of the workpiece.
	**Simulation programming mode:** The Simulation mode graphically depicts contours, tool movements and cutting processes. The working space, tools and chucking equipment are shown true to scale. During simulation, the CNC PILOT calculates the **machining and idle-machine times** for every tool. For lathes with several slides, the **synchronous point analysis** enables you to optimize your NC program.
	**TURN PLUS programming mode:** In TURN PLUS you describe the workpiece contour interactively in a graphic. If you then define the material and chucking equipment, the Automatic Working plan Generation (AWG) will generate the NC program automatically at a keystroke. As an alternative, you can create the working plan with the aid of interactive graphics (IAG).
	**Parameter organization mode:** The system behavior of the CNC PILOT is controlled by parameters. In this mode, you set the parameters to adapt the control to your situation. In addition, in this mode you describe the operating resources (tools and chucking equipment) and the cutting values.

Operating modes



Service organization mode: In Service you log in for password-protected functions, select the conversational language and make system settings. This operating mode also provides diagnostic functions for commissioning and checking the system.



Transfer organization mode: In Transfer, you exchange data with other systems, organize your programs and back-up your data.

The actual control is not accessible to the machinist. You should know, however, that your CNC PILOT stores all TURN PLUS and DIN PLUS programs that you enter on an integrated hard disk. This allows you to save a vast number of programs.

For data exchange and data backup, you can use the **Ethernet interface** and the USB memory medium. Data exchange on the basis of the **serial interface (RS-232)** is also possible.

1.3 Expansion Stages (Options)

The machine manufacturer configures the CNC PILOT according to the capabilities of the specific lathe. The following upgrades (options) are available, which enable you to adapt the control to your specific requirements:

TURN PLUS – Basis (ID 354 132-01):

- Graphically supported interactive contour definition
  - Graphic program entry for describing the workpiece blank and finished part
  - Geometry program for calculating and depicting contour points for which dimensional data are missing
  - Simple entry of standardized form elements such as chamfers, rounding arcs, undercuts, recesses, threads, fits
  - Simple handling of transformations such as datum shift, rotation, mirroring, duplication
- Graphically supported interactive DIN PLUS program generation
  - Individual selection of the operating mode
  - Selection of the tool and specification of cutting data
  - Direct graphic verification of the metal removing operation
  - Direct compensation capability
- Automatic DIN PLUS program generation
  - Automatic tool selection
  - Automatic generation of the working plan

TURN PLUS – C-axis option (ID 354 133-01):

- Depiction of programming in the XC plane (front/rear face) and ZC plane (unrolled lateral surface view)
- Hole pattern and figure pattern; any milled contours
- Interactive or automatic generation of the working plan, including C-axis machining

TURN PLUS – option for opposing spindles (ID 354 134-01):

- Rechucking with expert program
- Interactive or automatic generation of the working plan, including rechucking and machining the second setup

TURN PLUS – DXF import (ID 526 461-01):

- In TURN PLUS, read in the contours (blank and finished part contours, milled contours, contour trains) that are available in DXF format
- View and select DXF layers
- Load DXF contour into TURN PLUS

Opposing spindle—full-surface machining of a workpiece (ID 518 289-01):

- Spindle synchronism (G720)
- Parting control (G917, G991, G992)
- Traversing to a dead stop (G916)
- Mirroring and converting (G30)

In-process measurement—measuring on the machine (ID 354 536-01):

- With touch trigger probe
- For tool setup
- For workpiece measurement

Post-process measurement—measuring on external measuring stations (ID 354 537-01):

- Connecting the measuring equipment over the RS-232 interface
- Evaluating the measurement results in the NC program

Y axis (ID 354 138-01)

- Support of Y axis programming in DIN PLUS, TURN PLUS and in simulation
- Depiction of programming in the XY (front/rear face) and YZ (lateral surface view) planes
- DIN PLUS and TURN PLUS: Drilling, boring and figure patterns; any milling contours
- DIN PLUS: Cycles for drilling, boring and milling
- TURN PLUS: Interactive or automatic generation of the working plan, including Y-axis machining

B axis (ID 589 963-01)

- Support of B axis programming in DIN PLUS and in simulation
- The coordinate system is transformed to a tilted place in order to describe hole and figure patterns as well as any desired contours in the YZ plane
- Fixed cycles work in a tilted plane

Options can usually be retrofitted. Your machine manufacturer can give you more information.



This description covers all options. The operating sequences described in this manual may therefore deviate from those on your machine whenever a certain option is not supported by your system.

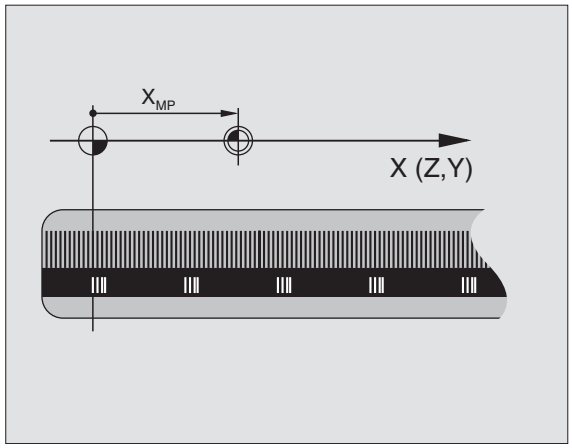
1.4 Fundamentals

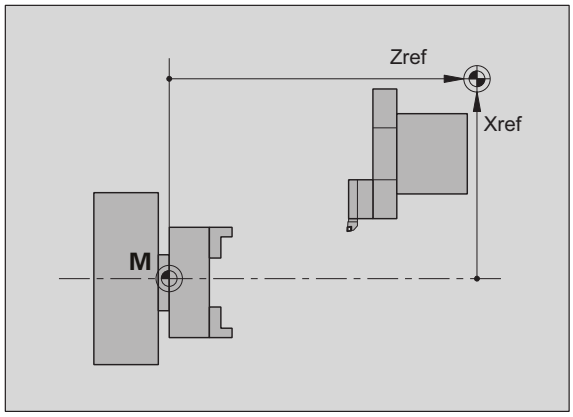
Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the slide or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the control when they are crossed over. From the signal the CNC PILOT can re-establish the assignment of displayed positions to machine positions. For linear encoders with distance-coded reference marks the machine axes need to move by no more than 20 mm, for angle encoders by no more than 20°.

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.





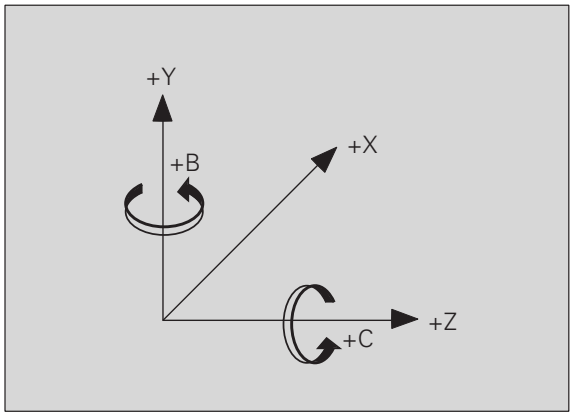
Axis designations and coordinate system

Coordinate system

The meanings of the coordinates X, Y, Z, B, C are specified in DIN 66 217.

The coordinates entered for the **principle axes** X and Z are referenced to the workpiece zero point. The angular data for the rotary axes B and C are given with respect to the zero point of the respective rotary axis.

On lathes, C axis movements are realized by turning the workpiece and B axis movements by tilting the tool (swivel head).



Axis designations

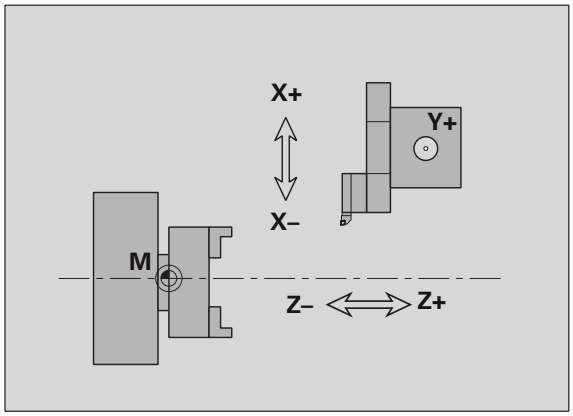
The cross slide is referred to as the **X axis** and the saddle as the **Z axis.**

All X-axis values that are displayed or entered are regarded as **diameters.** In TURN PLUS you can define whether the X axis values are diameters or radii.

Lathes with **Y axis**: The Y axis is perpendicular to the X axis and Z axis (Cartesian system).

When programming paths of traverse, remember to:

- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.



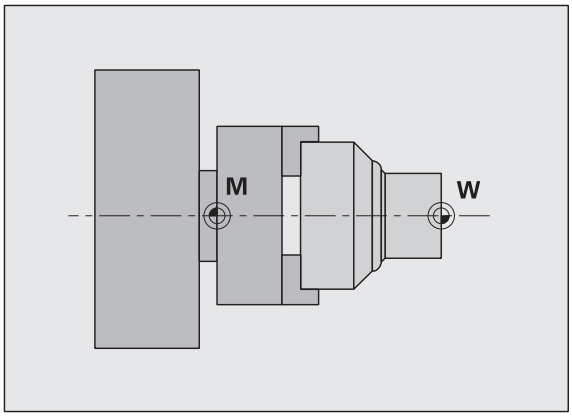
Machine reference points

Machine zero point

The point of intersection of the X and Z axes is called the **machine zero point.** On a lathe, the machine zero point is usually the point of intersection of the spindle axis and the spindle surface. The machine zero point is designated with the letter "M".

Workpiece zero point

For machining a workpiece, it is easier to reference all input data to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation. This point is the "workpiece zero point." The workpiece zero point is designated with the letter "W".



Absolute and incremental workpiece positions

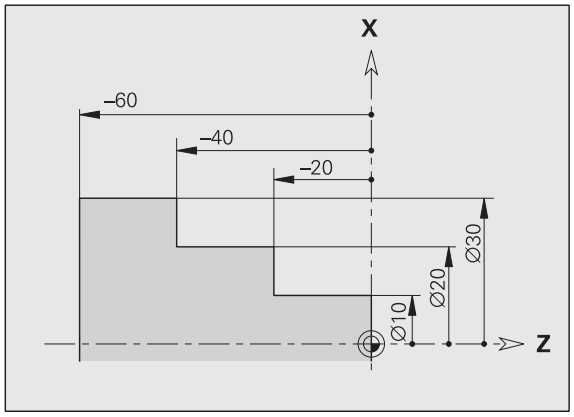
If the coordinates of a position are referenced to the workpiece zero point, they are referred to as **absolute coordinates.** Each position on a workpiece is clearly defined by its absolute coordinates.

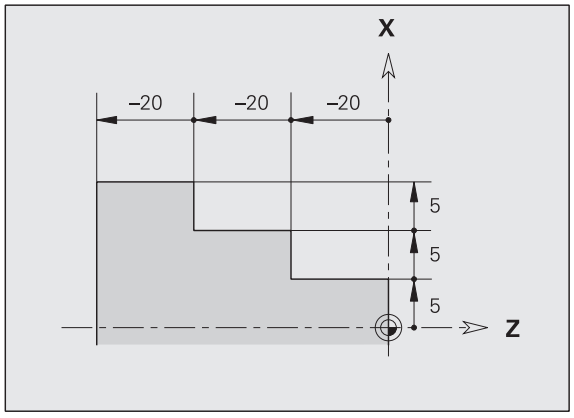
Incremental coordinates: Incremental coordinates are given with respect to the last programmed position. They specify the distance from the last active position and the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates.

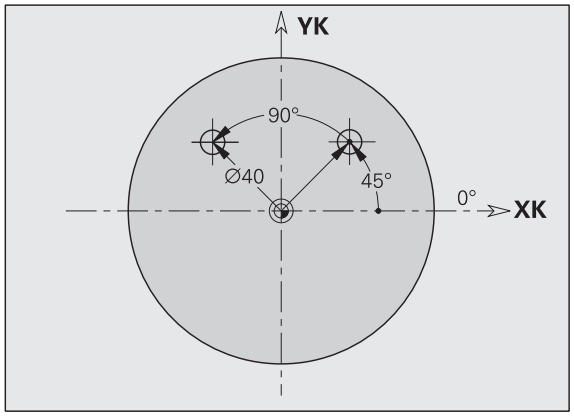
Absolute and incremental polar coordinates: Positions located on the face or lateral surface can either be entered either in Cartesian coordinates or polar coordinates.

When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for diameter and angle.

Absolute polar coordinates refer to the pole and the angle reference axis. Incremental polar coordinates always refer to the last programmed nominal position of the tool.







Units of measure

You can program the CNC PILOT either in the metric or inch system. The units of measurement listed in the table below apply to all inputs and displays.

Measure	Metric	Inches
Coordinates	mm	inches
Lengths	mm	inches
Angle	degrees	degrees
Shaft speed	rpm	rpm
Cutting speed	m/min	ft/min
Feed per revolution	mm/rev	inch/rev
Feed per minute	mm/min	inch/min
Acceleration	m/s^2	ft/s^2

1.5 Tool Dimensions

The CNC PILOT requires information on the specific tools for a variety of tasks, such as calculating the cutting radius compensation or the proportioning of cuts.

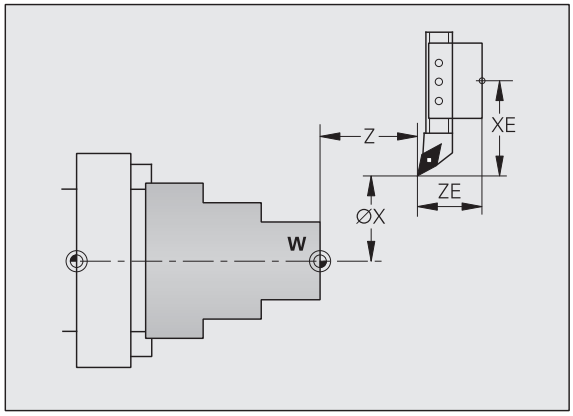
Tool dimensions: All position values that are programmed and displayed are referenced to the distance between the tool tip and workpiece zero point. Since the control only knows the absolute position of the tool carrier (slide), The CNC PILOT needs the dimensions XE and ZE to calculate and display the position of the tool tip, and for Y axis machining, it also needs the dimension in Y.

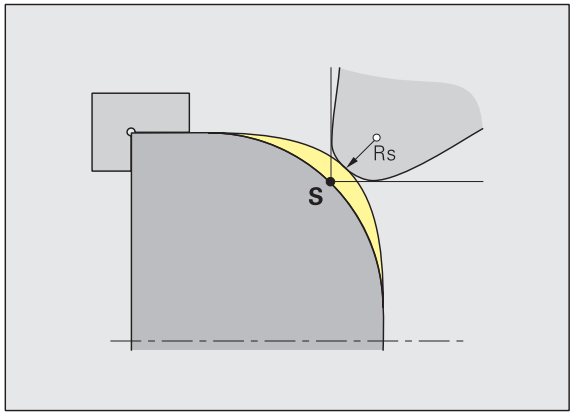
Tool compensation: The tool tip is subjected to wear during machining processes. To compensate for this wear, the CNC PILOT uses compensation values. The system automatically adds the compensation values to the values for length.

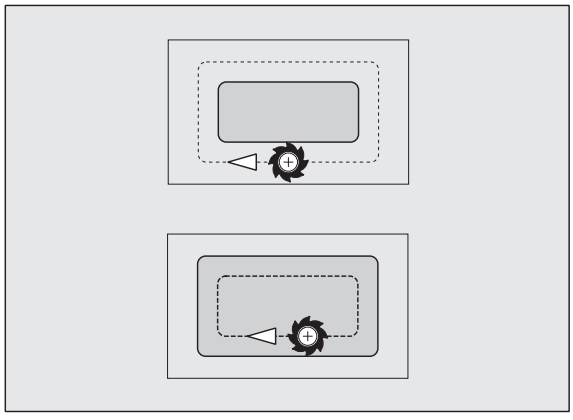
Tool radius compensation (TRC): The tip of a lathe tool has a certain radius. When machining tapers, chamfers and radii, this results in inaccuracies which the CNC PILOT compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical tool tip **S.** The TRC function compensates for this error by calculating a new path of traverse, the **equidistant line.**

Milling cutter radius compensation (MCRC): In milling operations, the outside diameter of the milling cutter determines the contour. When the MCRC function is not active, the system defines the center of the cutter as the zero point for paths of traverse. The MCRC function compensates for cutter radius by calculating a new path of traverse, the **equidistant line.**







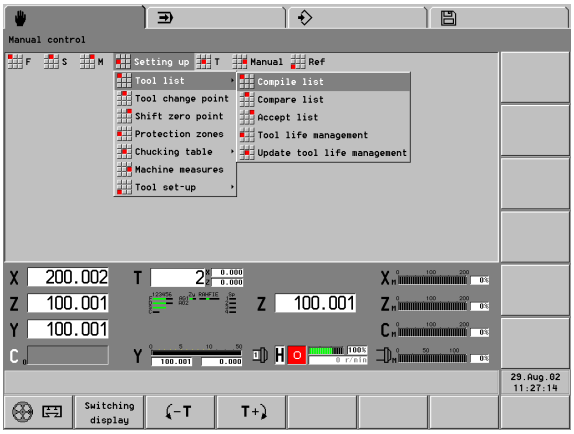


2

Basics of Operation

2.1 User Interface

Screen displays



1 **Operating mode bar:** Shows the status of the operating modes.
   - The active mode is shown with a dark-gray background.
   - Programming and organization operating modes:
     - The selected mode of operation is shown at right next to the symbol.
     - Additional information such as the selected program, submode, etc. are shown below the operating mode symbol.

2 **Menu bar and pull-down menu** enable you to select functions.

3 **Working window:** Size and content depend on the operating mode.

4 **Machine display:** Shows the current status of the machine (tool position, the cycle and spindle situation, active tool, etc.). The machine display is configurable.

5 **Status bar**
   - Simulation, TURN PLUS: Displays the current settings or information on the next operating steps.
   - Other operating modes: Displays the most recent error message

6 **Calendar date and service “traffic light”**
   - Date and time
   - A colored background signalizes an error or a PLC message.
   - The “traffic light” indicates the maintenance status of the machine.

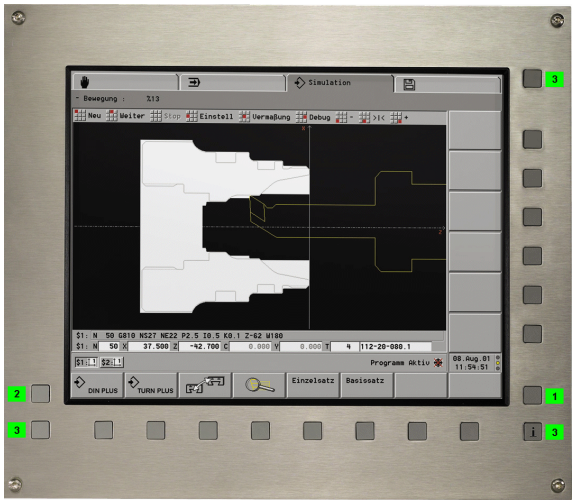
7 **Soft-key row:** Shows the current meaning of the soft keys.

8 **Vertical soft-key row:** Shows the current meaning of the soft keys. For more information: see the machine manual.

Controls and displays

Operating elements of the CNC PILOT:

- **Screen** with
  - Horizontal and vertical **soft keys:** The meaning is shown above or next to the soft keys.
  - Auxiliary key 1: Acts as the Esc key
  - Auxiliary key 2: Acts as the Insert key
  - Auxiliary keys 3: PLC keys
- **Keyboard** with
  - **Alphabetic keyboard** with integrated **numeric keypad**
  - Keys for **operating mode selection**
  - **Touchpad:** For cursor positioning (menu or soft key selection, selection from lists, selecting edit boxes, etc.)
- **Machine operating panel** with
  - Operating elements for the manual and automatic operation of the lathe (cycle keys, manual direction keys, etc.)
  - **Handwheel** for exact positioning in manual mode
  - **Override button** for feed-rate override





Operating information on the touchpad: Normally you use the touchpad as an alternative to the cursor keys. In the following, the keys below the touchpad are referred to as the left and right mouse keys.

The functions and operation of the touchpad are similar to the mouse operation of the Windows operating systems.

- Single click of the left mouse key or tap on the mouse pad:
  - Positions the cursor in lists or input windows.
  - Activates menu points, soft keys or buttons.
- Double click of the left mouse key or double tap on the mouse pad:
  - Activates the selected element in lists (activates the input window).
- Single click of the right mouse key:
  - Same function as the Esc key. Prerequisite: the ESC key is permitted in this situation (for example to go back by one menu level).
  - Same function as the left mouse key when selecting soft keys or buttons.

Selecting the operating mode

Keys for operating mode selection	
	**Manual control operating mode**
	**Automatic operating mode**
	**Programming modes**
	**Organization modes**

You can usually switch operating mode at any times. In some situations, you cannot switch operating modes when a dialog box is open. In this case, close the dialog box before changing operating modes. After the change, the new mode starts in the function in which it was last exited.

In the **Programming and Organization operating modes,** the CNC PILOT differentiates between the following situations:

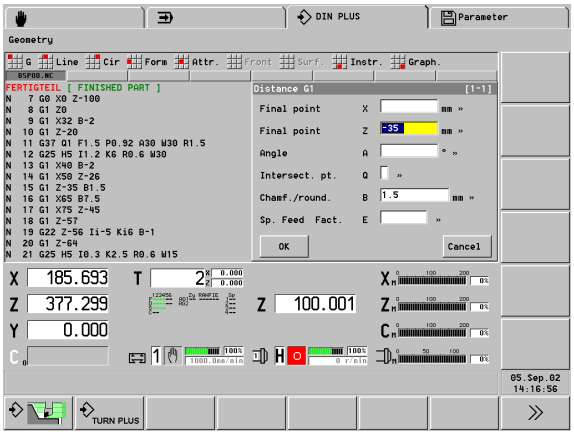
- No operating mode is selected (no entry next to the operating mode symbol): Select the desired mode from the menu.
- Operating mode selected (indicated next to the operating mode symbol): The functions of this operating mode are available.
- Within the programming or organization modes, you can switch the modes by soft key or by repeatedly pressing the corresponding mode key.

Data input, selection of functions

Data are entered and edited in **input windows.** An input window consists of several **input fields.** You position the cursor with the touchpad or with the PgUp/PgDn keys to the input box.

Once the cursor is located in the box, you can enter your data or overwrite existing data. With the right/left arrow keys you can place the cursor on a position **within** the input box in order to delete characters or add data. The “up/down” arrow keys or “Enter” confirm and terminate the entry.

Some dialogs have more input fields than a window can show. In these cases, more than one input window appears on the screen, one superimposed on the other. You will recognize this through the window number in the top line. To toggle between input windows, use the PgUp/PgDn keys.



When you press the OK button, the control accepts the data entered or edited. As an alternative you can press the Ins key to confirm the data, regardless of the cursor position. If you leave the input window by pressing the "Cancel" button or the ESC key, entries or changes will be lost.

If the dialog consists of more than one input window, you already confirm the data when pressing the PgUp/PgDn key.



Note: Instead of selecting the OK or Cancel button, you can press the Ins or Esc key.

List operations: DIN PLUS programs, tool lists, parameter lists, etc. are displayed as lists. You can scroll through a list with the touchpad or arrow keys to check data, to select the position where you wish to enter data, or to highlight items for operations like deleting, copying, editing, etc.

After having selected the desired list position or a list item, press the ENTER, INS, ALT or DEL key to execute the operation.

Menu selection: The individual menu items are preceded by the 9-field symbol with one field marked. This key represents the key on the numeric keypad. Press the marked button to select the function.

The function selection begins in the horizontal menu row, then goes to the pull-down menus. In the pull-down menu you again press the marked key. As an alternative you can select the menu item with the touchpad or with the Up/Dn arrow keys and then press ENT.

Soft keys: The meaning of the soft keys is dependent on the current operating situation. The CNC PILOT indicates the function of the soft keys with symbols or keywords.

Some soft keys work like "toggle switches." A function is active when the associated field in the soft-key row is highlighted in color. The setting remains in effect until the function is switched off.

Screen buttons: Example of screen buttons are the OK and Cancel buttons for terminating a dialog box or the buttons contained in the "Extended inputs" window.

Select the button by cursor and press ENT, or select the button using the touchpad and press the left mouse button.

2.2 Info and Error System

The info system

The info system calls excerpts from the User's Manual to the screen. The header shows the selected topic.

You'll usually find information on the current operating situation (context-sensitive help). if no context-sensitive help is available for a specific situation look for the topics in the following sources:

- The table of contents
- The subject index
- The search functions

Cross references are marked in the text. Click the cross references by touchpad to jump to that topic.

Calling and exiting the info system:



▶ Call the info system



▶ Exit the info system

Contents, subject index, search function: When called, the info system opens the standard window (figure at upper right). Use the soft key to switch to the Content/Index window in order to find topics through the table of contents or subject index, or through the search function (figure at lower right).



Content/Index window:

▶ Soft key active: Window is displayed.

▶ Soft key not active: Window is not displayed.

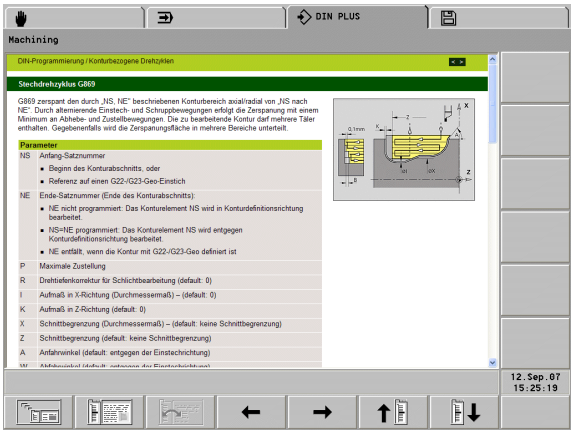
Size of the info window: Use the soft key to switch the window to maximum size.

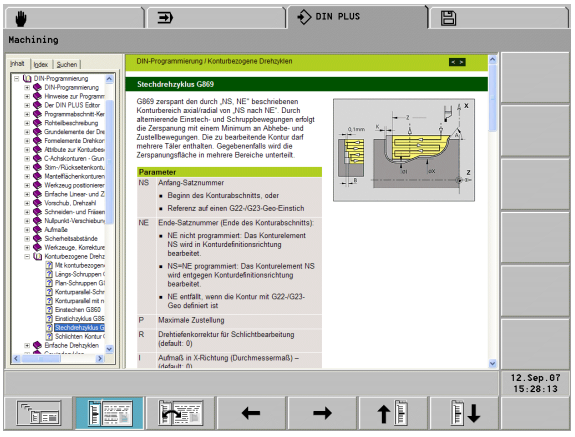


Large window or standard window:

▶ Soft key active: The info is displayed in the large window.

▶ Soft key not active: The info is displayed in the standard window.





Navigating in the info system:

▶ You navigate by touchpad as is usual in Windows programs.

If the topic of information exceeds the window size:

▶ Navigate with the up/down cursor keys and PgUp/PgDn keys through the displayed topic. Prerequisite: The cursor must be located in the topic window and not in the Content/Index window.



Moving the cursor:

▶ Press the soft keys. The cursor switches between the topic window and the Content/Index window.

Next/previous topic:

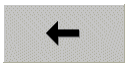


▶ Call the next topic from the table of contents.

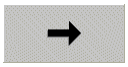


▶ Call the previous topic from the table of contents.

Next/previous topic: The info system saves the history.



▶ Go to the previous topic.



▶ Go to the next topic.

OEM help: This soft key is only operable if the machine tool builder has saved information in the online help.



▶ Call the OEM help.

Context-sensitive help

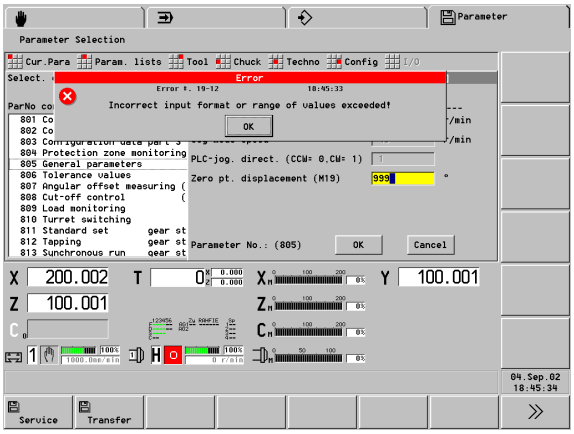
You'll usually find information on the current operating situation (context-sensitive help). if no context-sensitive help is available for a specific situation look for the topics in the following sources:

- The table of contents
- The subject index
- The search functions

Direct error messages

The CNC PILOT uses a direct error message whenever immediate error correction is possible. Confirm the message and correct the error.

Example: The input value of the parameter is out of range.



Information of the error message:

- **Error description:** Explains the error
- **Error number:** For service questions
- **Time of day:** Time when the error occurred (for your information)

Symbols

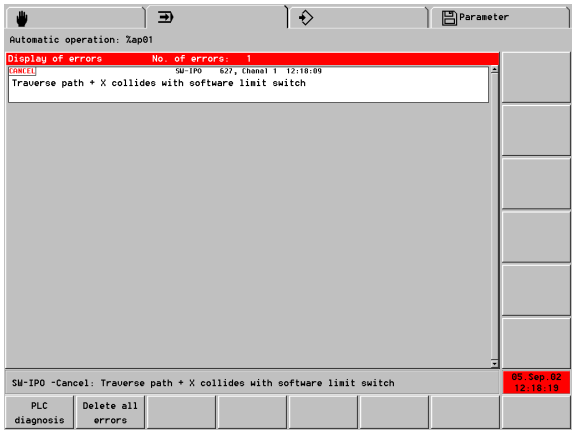
Warning: The CNC PILOT indicates the problem. The program run / operation continues.



Error: The program run/operation is stopped. You must correct the error before you can continue the current job.



Error display



If during the system start or program run or other operation an error occurs, it is indicated in the date box, displayed in the status line, and saved in the error display.

The date and time remain highlighted in red until all of the errors have been canceled.

Information of the error message:

- **Error description:** Explains the error
- **Error number:** For service questions
- **Channel number:** Slide for which the error occurred.
- **Time of day:** Time when the error occurred (for your information)
- **Error class** (only with errors):
  - **Background:** This message serves as information, or merely a “small” error has occurred.
  - **Cancel:** The current operation (cycle run, traverse command, etc.) was cancelled. You can resume operation once the error has been cleared.
  - **Emergency stop:** All traverse and the execution of the DIN program was stopped. You can resume operation once the error has been cleared.
  - **Reset:** All traverse and the execution of the DIN program was stopped. Switch off the control for a moment, then restart. Contact your machine manufacturer if the error occurs again.

System error, internal error: If a system error or internal error occurs, write down all information on the displayed message and inform your machine manufacturer. You cannot correct an internal error. Switch off the control and restart.

Warnings during the simulation: In the event of problems during simulation of an NC program, the CNC PILOT displays a warning in the top line.

Viewing and deleting error messages:



▶ Activate the error display. The error system shows all accumulated errors.

▶ If more than one error is shown, navigate with the cursor keys within the error display.

DEL

▶ Deletes the error message marked with the cursor.

Delete all errors

▶ Deletes **all** error messages.



▶ Display further information on the error marked with the cursor.

ESC

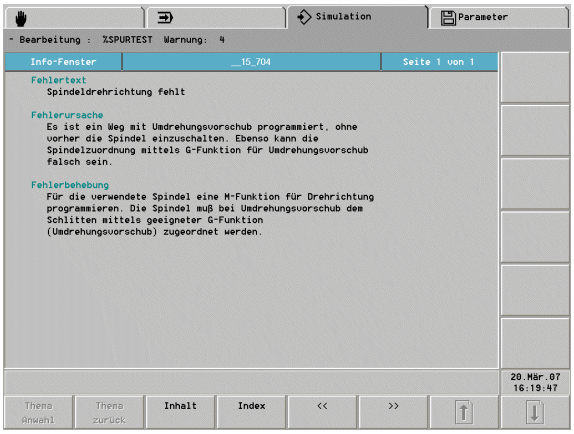
▶ Exit the error display.

Additional information on error messages

When an error message occurs, press the info key, or place the cursor on the error message in the error display and then press the info key, to get further information on the respective error.

Meaning of the soft keys:

>>	▶ Information on the next error message.
<<	▶ Information on the previous error message.
Content	▶ Switches to the general info system
Index	▶ Switches to the general info system



PLC display

The PLC window is used for PLC messages and the PLC diagnosis. Your machine manual provides more detailed information on the PLC window.

Activating the PLC display:

ϟ	▶ Opens the “error display”
PLC diagnosis	▶ Switches to the PLC window
ESC	▶ Close PLC window
CNC diagnosis	▶ Return to error display

The PLC window is shown as alternative to the error window.



2.3 Data Backup

The CNC PILOT stores NC programs, operating-resource data and parameters on the hard disk. Since the possibility of damage to the hard disk due to excessive vibration or shock cannot be eliminated, HEIDENHAIN recommends making regular backup copies of your programs, operating resource data and parameters on a PC or on USB memory media.

You can use DataPilot 4290, the WINDOWS "Explorer" or other suitable programs for backing up your data on a PC.

For data exchange and data backup, you can use the **Ethernet interface** and the USB interface. Data exchange on the basis of the **serial interface (RS-232)** is also possible.

2.4 Explanation of Terms

- **MP:** With machine parameters (MP) the control is interfaced to the machine, settings are made, etc.
- **Cursor:** In lists, or during data input, a list item, an input field or a character is highlighted. This "highlight" is called a cursor.
- **Cursor keys:** You can move the cursor with the arrow keys, PgUp, PgDn of the touchpad.
- **Navigating:** Within a list or an input box, you can move the cursor to any position you would like to check, change, delete or add to. In other words, you "navigate" through the list.
- **Active/inactive functions, menu items:** Functions or soft keys that at present cannot be selected are shown in gray.
- **Dialog box:** Dialog boxes are also called input windows.
- **Editing:** Editing is changing, deleting and adding to parameters, commands, etc. within programs, tool data or parameters.
- **Default value:** If the parameters of DIN commands or other parameters are preassigned values, these values are referred to as "default values."
- **Byte:** The capacity of a storage disk is indicated in bytes. Since the CNC PILOT features a hard disk, the individual program lengths (file sizes) are expressed in bytes.
- **Extension:** File names consist of the actual name and the extension. The name part and the extension part are separated by a dot ".". The extension indicates the type of file. Examples:
  - „*.NC"DIN programs
  - „*.NCS"DIN subprograms
  - „*.MAS"Machine parameters



3

Manual Control and Automatic Modes

3.1 Switch-On, Switch-Off, Reference Run

Switch-on

In the header, the CNC PILOT displays the individual steps of the system start and then prompts you to select an operating mode.

Whether the reference run is necessary depends on the encoders installed:

- EnDat encoder: Reference run is not necessary.
- Distance-coded encoders: The position of the axes is ascertained after a short reference run.
- Standard encoder: The axes move to known, machine-based points.

After completion of the reference run:

- This position display is activated.
- Automatic mode can be selected.

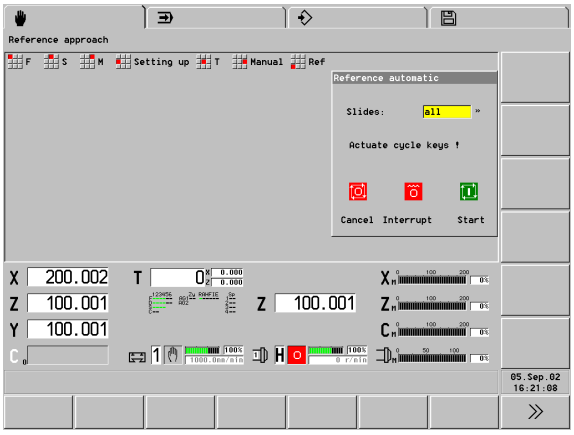
The **software limit switches** are active only after you have traversed the reference marks.

Reference run for all axes

Select Ref > Reference automatic.

The “Status of reference run approach” dialog box informs you of the current status.

Either set the slides that need to find a reference or set “All slides” (“reference automatic” dialog box)



To start the reference run, press Cycle Start.

“Feed stop” interrupts the reference run. Cycle start resumes the reference run.

“Cycle stop” interrupts the reference run.

The **sequence** in which the axis make their reference run is defined in MPs 203 and 253 ff.

Reference jog for single axis

Select Ref > Reference jog.

The “Status of reference run approach” dialog box informs you of the current status.

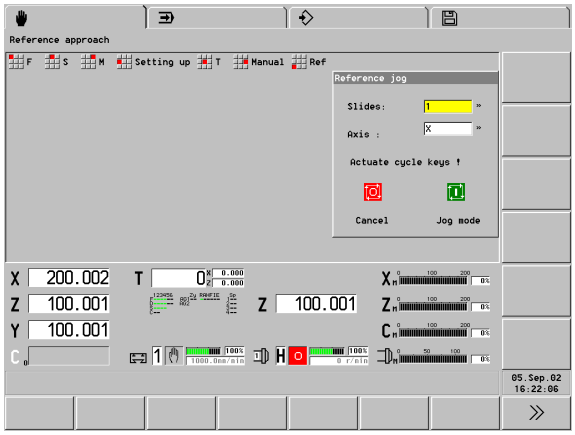
Set slides and axes (“Reference jog” dialog box)



The reference run is conducted for as long as the Cycle Start key stays pressed. To interrupt the reference run, release the key.



“Cycle stop” interrupts the reference run.



Monitoring EnDat encoders

If your machine is equipped with EnDat encoders, the control saves the axis positions during switch-off. During switch-on, the CNC PILOT compares for each axis the position during switch-on with the position saved during switch-off.

If there is a difference, one of the following messages appears:

- “Axis was moved after the machine was switched off.” Check the current position and confirm it if the axis was in fact moved.
- “Saved encoder position of the axis is invalid.” This message is correct if the control has been switched on for the first time, or if the encoder or other control components involved were exchanged.
- “Parameters were changed. Saved encoder position of the axis is invalid.” This message is correct if configuration parameters were changed.

The cause for one of the above listed messages can also be a defect in the encoder or control. Please contact your machine supplier if the problem recurs.

Switch-off

"Shutdown" is available in the programming and organization modes if no operating mode is selected.

Shutdown

- Press the soft key to switch off the CNC PILOT.
- Confirm the security query with OK. After a few seconds, the CNC PILOT requests you to switch off the machine.

Proper switch-off is recorded in the error log file.



3.2 Manual Control Mode

The Manual Control mode offers various functions for setting up the machine, for measuring tool dimensions and for manually machining workpieces.

Options of operation:

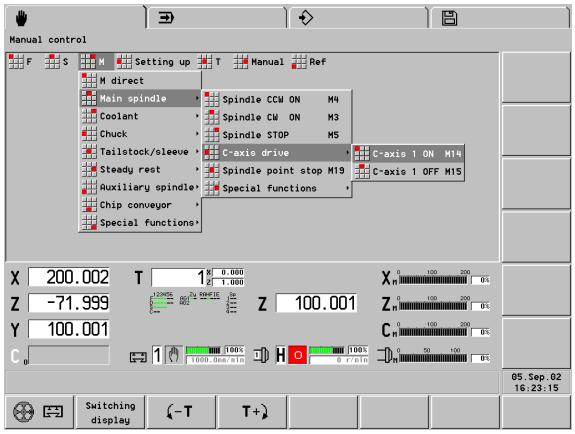
- **Manual mode:** With the "machine keys" and the handwheel, you can control the spindle and move the axes to machine the workpiece.
- **Setup mode:** Here you enter the tools to be used, set the workpiece datum, the workpiece change point, the protection zone dimensions etc. In this way you prepare the machine to machine the workpieces.
- **Determining the tool dimensions:** You can find the tool dimensions by touching the workpiece with the tool or with a touch probe. As an alternative you can measure the dimensions with a gauge and enter them in the tool database.

For manual control you can configure the **machine display** for up to six variants (see "Machine Display" on page 97). You adjust by soft key which variant will be displayed.

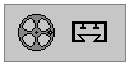
In Manual Control mode, the data are entered and displayed according to control parameter 1 either in **meters** or in **inches.**

Remember: If the machine has **not been referenced:**

- The position display is not valid.
- The **software limit switches** are not active.



Soft keys for manual and setup functions

	■ Assign a handwheel to an axis ■ Set the handwheel transmission
Switching display	Switch the machine display
T-	Turret one position backward
T+	Turret one position forward
F	Enter the feed per revolution
S	Spindle speed, entering
M	Enter the M function

Entering the machine data

Setting the feed rate

In the menu group F you define a feed rate per revolution or per minute.

Setting the feed per revolution:

- Select F > Feed per revolution
- Enter the feed rate in mm/rev (or inches/rev)

Setting the feed rate per minute:

- Select F > Feed per minute
- Enter the feed rate in mm/min (or inches/min) and press OK.

Setting the spindle speed or spindle position

In the menu group S, you define the spindle speed, a constant cutting speed, or you position the spindle.

Setting the spindle speed:

- Select S > Spindle speed S
- Enter the speed in rpm

Setting the constant surface speed:

- Select S > V constant.
- Enter the cutting speed in m/min (or ft/min) and press OK.



You can enter a constant cutting speed only for slides with an X axis.

Entering the speed limitation

Available as of software version 625 952-05.

Precondition: Log on as “System manager” (or higher).

- To switch to the required spindle, press the spindle change key
- Select S > Spindle point stop
- Enter the speed in rpm

The current speed limitation for the selected spindle is entered as proposed value. The speed limitation entered is saved in MP 805, .. (absolute maximum rpm).

Conduct a spindle point stop (spindle positioning):

- To switch to the required spindle, press the spindle change key
- Select S > Spindle point stop
- Enter the angular position (“spindle point stop” dialog box)



- Cycle Start positions the spindle



- “Cycle stop” exits the dialog box

Insert the nominal tool

▶ Select T. Enter the turret position, or



▶ the next turret position, or



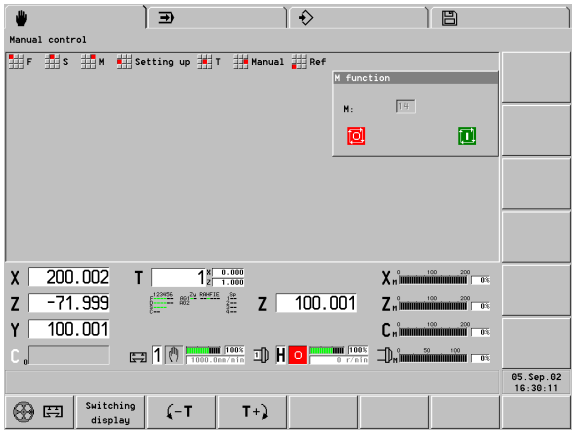
▶ the previous turret position, or

Functions of the tool change:

- Move the tool into position
- Calculate “new” tool dimensions
- Display “new” actual values in the position display

M commands in Manual Control mode

In the menu group M you either type in the M function to be run or you select the desired function from the menu.



To run an M function:

▶ Select M > M direct

▶ Enter the M number (“M function” dialog box)



▶ Cycle Start runs the M function



▶ “Cycle stop” exits the dialog box

To select and run an M function:

▶ Select “M function”

▶ Using the menu, select the M function



▶ Cycle Start runs the M function



▶ “Cycle stop” exits the dialog box



The content of the M menu depends on the machine. Yours may differ from the example shown here.

Manual turning operations

The “manual” menu group includes G functions, simple longitudinal and transverse turning, and manual NC programs written by the machine tool builder.

Simple longitudinal and transverse turning operations:

- Select Manual > Constant feed
- Select the direction of feed (“Constant feed” dialog box)
- Control the feed rate with the cycle keys



With “constant speed,” a feed rate per revolution must be defined.

To run a G function:

- Select Manual > G function
- Enter the G number (“G function” dialog box)
- Enter the function parameters
- Press OK: The G function is executed

The following G functions are permitted:

- G30 – Rear-face machining
- G710 – Adding tool dimensions
- G602..G699 – PLC functions

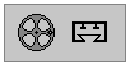
Manual NC programs
Depending on the configuration of a lathe, the machine manufacturer can includes NC programs supporting the machinist in manually operating the lathe (Example: Switching to rear-face machining).

- Select “Manual”
- Select the desired Manual NC program from the menu
- The control loads the NC program and displays it

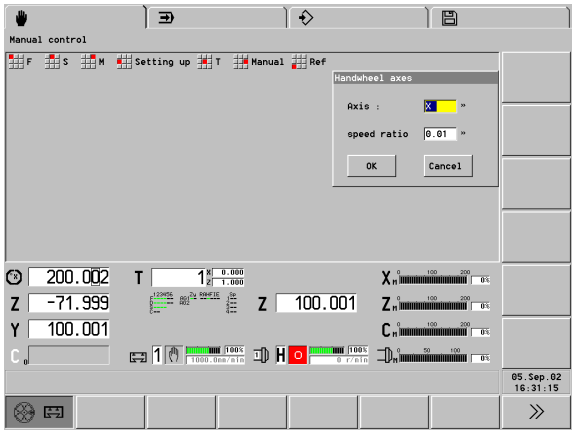


- Cycle Start activates the NC program

Handwheel



- Assign the handwheel to a principal axis or C axis (“Handwheel axes” dialog box
- Define the feed rate or angle of rotation per handwheel increment (“Handwheel axes” dialog box).
- **The cancel the handwheel assignment,** press the “Handwheel” soft key with opened dialog box.



The handwheel assignment and speed ratio are shown in the machine display (the axis letter and the decimal place of the handwheel traverse ratio are marked).

The handwheel assignment is canceled by:

- Switching the slides
- Switching the operating modes
- Pressing a manual direction key

Spindle and manual direction keys

The machine operating keys are used for machining under manual control and for special functions such as finding positions or compensation values (Teach-in, scratching, etc.).

The tool is activated and the spindle speed and feed rate defined first.

You define the following parameters by MP:

- MP 805, 855, ...: Spindle speed during jog
- MP 204, MP 254, ...rapid traverse velocity



To move the slide diagonally, press the X and Z axis direction keys simultaneously.

Spindle keys	
	Switch the spindle on in M3/M4 direction.
	The spindle rotate in M3/M4 direction as long as the key is pressed (spindle jog)
	Spindle STOP

Manual direction keys (jog keys)	
	Move slide in X direction
	Move slide in Z direction
	Move slide in Y direction.
	To move the slide in rapid traverse, simultaneously press the rapid traverse key and the axis direction key

Slide/spindle change key

On lathes with **multiple slides** the following keys, functions and displays refer to the selected **slide:**

- Manual direction keys
- Setup functions (for example workpiece zero point setting, set tool change point)
- Slide-dependent display elements of the machine display
- Display of the “selected slide”: Machine display

The “selected slide” is listed in the “slide display” (see “Machine Display” on page 97).

To change the slide, press the slide change key

On lathes with **multiple spindles** the following keys and displays refer to the selected spindle:

- Spindle keys
- Spindle dependent display elements of the machine display

The “selected spindle” is listed in the “spindle display” (see “Machine Display” on page 97).

To change the spindle, press the spindle change key

Slide/spindle change key	
	Switch to the next slide
	Switch to the next spindle

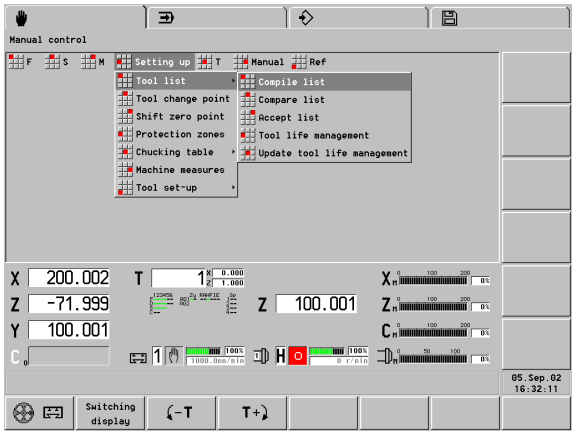
3.3 Table for Tools and Chucking Equipment

The tool list (turret table) indicates the current tool carrier assignment. To set up a tool list, enter the ID numbers of the tools.

You can use the entries in the TURRET section of the NC program to set up the tool list. The “Compare list” and “Accept list” functions refer to the NC program last interpreted in automatic mode.

Danger of collision

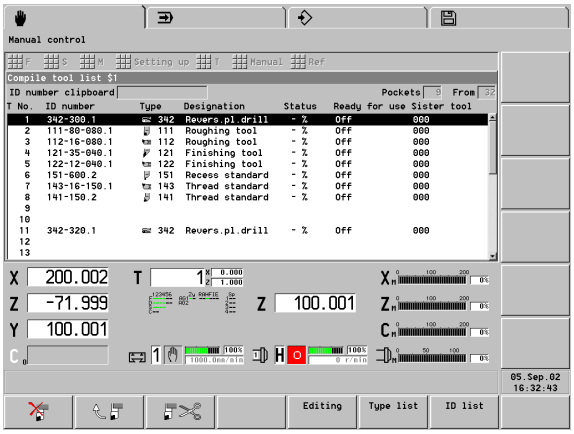
- Compare the tool list with the tools actually in the tool magazine and check the tool carrier **before** executing the part program.
- The tool list and the dimensions of the registered tools must correspond to the tool actually present, because the CNC PILOT uses this data for all slide movements, protective zone monitoring, and other slide movements.



Soft keys for setting up the tool list	
	Deleting a tool
	Take the tool from the clipboard
	■ Deleting a tool ■ Place the tool in the clipboard
Editing	Edit the tool parameters
Type list	Entries in the database – by tool type
ID list	Entries in the database – by ID number

Setting up a tool list

In "Setup tool list" you can declare the tool list independently from the data of an NC program.


Manual control
Compile tool list $1
ID number clipboard
Pockets 0 From 32
T No. ID number Type Designation Status Ready for use Sister tool
1 342-300.1 342 Revers.pl.drill - % Off 000
2 111-80-080.1 111 Roughing tool - % Off 000
3 112-16-080.1 112 Roughing tool - % Off 000
4 121-35-040.1 121 Finishing tool - % Off 000
5 122-12-040.1 122 Finishing tool - % Off 000
6 151-600.2 151 Recess standard - % Off 000
7 143-16-150.1 143 Thread standard - % Off 000
8 141-150.2 141 Thread standard - % Off 000
11 342-320.1 342 Revers.pl.drill - % Off 000
X 200.002
Z -71.999
Y 100.001
T 1
Z 100.001
05.Sep.02
16:32:43
Editing
Type list
ID list

Entering a tool

Select Setting up > Tool list > Compile list

Select the tool location.

Entering the tool directly:

Press ENTER (or the INS key). The CNC PILOT opens the "Setup" dialog box

Enter the ID number and click OK to close the dialog box.

Select the tool from the database:

Type list	List the tools by type mask, or
ID list	List the tools by ID number mask
	Place the cursor on the nominal tool.
Insert	Insert the tool.

Deleting a tool

Select Setting up > Tool list > Compile list

Select the tool location.



Use the soft key, or

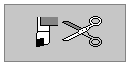


Press the DEL key to delete the tool

Changing the tool location

Select Setting up > Tool list > Compile list

Select the tool location.



Deletes the tool and saves it in the “ID number clipboard”

Select a new tool location.



Take the tool from the “ID number clipboard” If the location was occupied, the previous tool is taken into the clipboard.

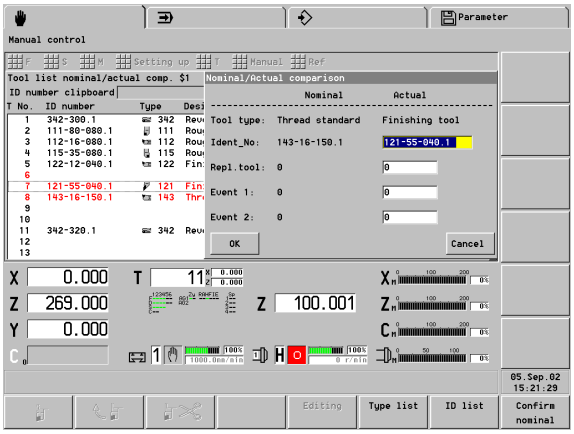
Comparing a tool list with an NC program

The CNC PILOT compares the current tool list with the entries in the NC program last translated in automatic mode. The entries in the TURRET section are considered to be **nominal tools.**

The CNC PILOT shows the following tools marked:

- Actual tool not equal to nominal tool
- Actual tool not occupied, nominal tool occupied

Tool locations that are not assigned in the NC program cannot be selected.



Danger of collision

- Tool pockets that are occupied but, according to the NC program, are **not** needed, are not marked.
- The CNC PILOT accounts for a tool entered, even if it does not correspond to the nominal assignment.

Comparing a tool list

Select Setting up > Tool list > Compare list

The CNC PILOT shows the current contents of the tool list and marks deviations from the programmed tool list.

Select **marked** tool pocket

Ins

Press the INS key (or ENTER). The CNC PILOT opens the “nominal-actual comparison” dialog box

Confirm nominal

Accept the ID number of the “nominal tool” in the tool list

Type list

ID list

Look for the tool in the database

Insert

Insert the tool.

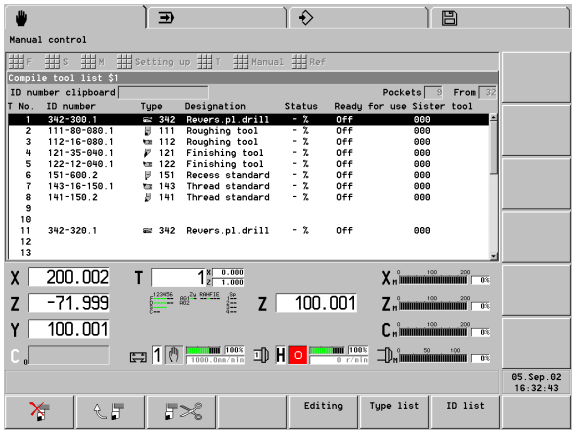
Transferring the tool list from an NC program

The CNC PILOT transfers the new tool assignment from the TURRET section (reference: the NC program last interpreted in Automatic mode).

Depending on the previous turret assignment, the following might occur:

- **Tool not used:** The CNC PILOT enters the new tools in the tool list. Positions that were occupied in the old tool list, but are not used in the new list, are retained. Delete the tool, if required.
- **Actual tool location differs from location in tool list:** A tool is **not** entered when its newly assigned pocket differs from the pocket specified in the tool list. The CNC PILOT displays a message indicating this error. Change the tool location.

As long as a tool position differs from the nominal assignment it remains highlighted.


Manual control
Compile tool list S1
ID number clipboard
Pockets
From
T No. ID number Type Designation Status Ready for use Sister tool
1 342-300.1 342 Revers.pl.drill - % Off 000
2 111-80-080.1 111 Roughing tool - % Off 000
3 112-16-080.1 112 Roughing tool - % Off 000
4 121-35-040.1 121 Finishing tool - % Off 000
5 122-12-040.1 122 Finishing tool - % Off 000
6 151-600.2 151 Recess standard - % Off 000
7 143-16-150.1 143 Thread standard - % Off 000
8 141-150.2 141 Thread standard - % Off 000
11 342-320.1 342 Revers.pl.drill - % Off 000
X 200.002
Z -71.999
Y 100.001
T 1
Z 100.001
05.Sep.02
16:32:43
Editing
Type list
ID list



Danger of collision

- Tool locations that are occupied but, according to the NC program, are not needed, are kept.
- The CNC PILOT accounts for a tool entered, even if it does not correspond to the nominal assignment.

Accepting the tool list

Select Setting up > Tool list > Accept list

Simple tools

The setup functions use tools that are listed in the database. If the NC program uses “simple tools,” proceed as follows:

- Run an interpretation of the NC program. The CNC PILOT automatically updates the tool list.
- If the positions in the tool list are occupied by “old” tools, the confirmation request “Update tool list?” appears. The entries are updated only after you have confirmed the request.

Tools that are not registered in the database are identified by the code “_AUTO_xx” (xx: T number), and not by an ID number.



Define the parameters of simple tools in the NC program.

Tool life management

The tool life management allows you to define the **sequence of exchange** and declare the tool to be ready for use. The tool life/ quantity is defined in the tool database.

Apart from ID numbers and tool type descriptions, the tool list includes data for tool life management:

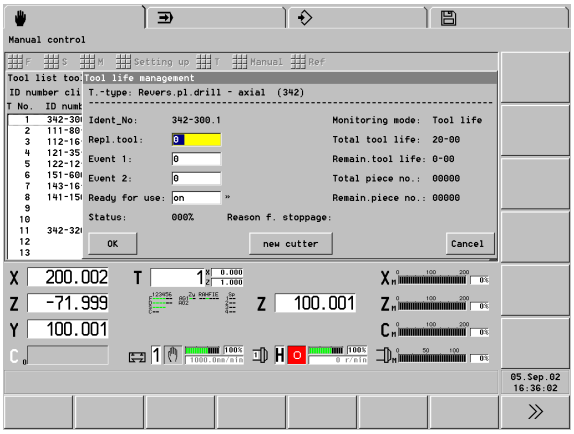
- **Status:** Shows the remaining tool life/quantity.
- **Ready for use:** When the tool life has expired/the defined number of parts has been produced, the tool is "not ready for use" any longer.
- **Atw (replacement tool):** If a tool is "not ready for use," a replacement tool is inserted.

The "Tool life management" dialog box is used for entering and displaying the tool life data.

You can use the variable programming in your NC program to evaluate sequential events that you enter in "Event 1" and "Event 2."

Tool life management parameters:

- **Repl. tool** (replacement tool): T number (turret position) of the replacement tool
- **Event 1:** Sequential event that is triggered when the life of a tool has expired/a tool has produced the defined quantity (Event 21..59).
- **Event 2:** Sequential event that is triggered when the tool life/ quantity of parts of the "last tool" has expired/a tool (Event 21..59).
- **Ready for use:** Set the tool to "ready for use" or "not ready for use" (applies to tool life management only).



The tool life data are evaluation only if the tool life management is **active.**

Entering the tool life parameters

Select Setting up > Tool list > Tool life management

The CNC PILOT displays the entered tools

Select the tool location.

Press ENTER. The CNC PILOT opens the “Tool life management” dialog box.

Enter the replacement tool and the other tool life parameters.

Press the “New cutting edge” button: The CNC PILOT sets the tool life/quantity to the value programmed in the database and sets the tool to **ready for use.**

Updating the tool life data of all tools

Select Setting up > Tool list > Update tool life management

Confirm the confirmation request with OK; the CNC PILOT sets the tool life/quantity to the value defined in the database and sets all tools in the tool carrier to **ready for use.**

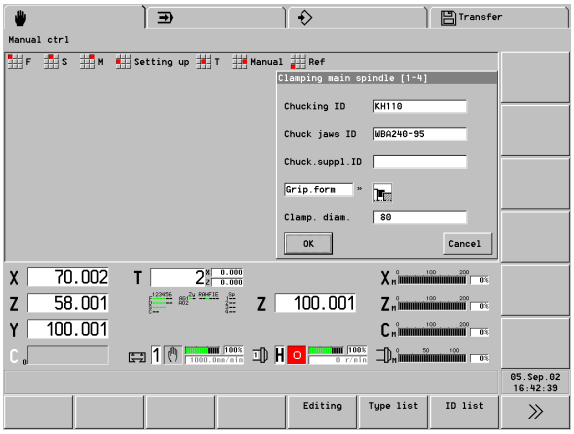
The CNC PILOT displays the “Tool list – Tool life management” for inspection.

Application example: The cutting edges of all tools used have been replaced. Part production is to be continued, using the tool life management function.

Setting up the chucking table

The **chucking table** is evaluated by the concurrent graphics.

To switch to the chucking assignment of further spindles, press the PgUp/PgDn keys.



Parameters for “spindle x” (main spindle, spindle 1, ..)

- Chucking ID: Reference to database
- Chuck jaws ID: Reference to database
- Chuck supplement ID: Reference to database
- Clamp form: Define the inside/outside clamp and clamp step used
- Clamping diameter: The diameter at which the workpiece is clamped. (Workpiece diameter when clamped externally; inside diameter when internal clamping is used)

“Tailstock” parameters

- Sleeve center ID: Reference to database

Setting up the chucking table

Select Setting up > Chucking table > Main spindle (or Tailstock)

For chuck, chuck jaws and chuck supplement: Enter the ID number of the chucking equipment.

Type list	List the chucking equipment by type mask
ID list	List the chucking equipment by ID number mask

Select the chucking equipment from the database

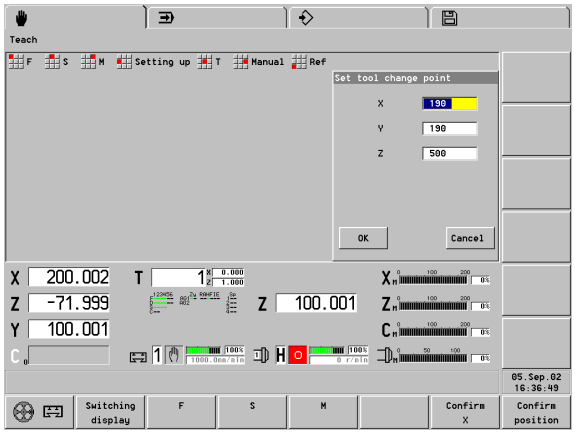
>>	**Clamp form:** Press the soft key several times to set the clamp form

Enter the clamping diameter

3.4 Setup Functions

Setting the tool changing point

With G14, the slide moves to the **tool change point.** Always program the tool change point as far from the workpiece as possible to allow the turret to rotate to any position.



The tool change point is a setup parameter.

Setting the tool changing point

For more than one slide: Define the desired slide

Select Setting up > Tool change point

The “tool change point” dialog box indicates the valid position.

Enter the position of the tool change point.

Teach in the tool change position

Confirm position

Move the slide to the “tool change point.”

Take over the tool change point, or

Confirm Z

Move the axis to the tool change point (or X or Y axis).

Take over the position of the axis

Shifting the workpiece zero point

- The shift is referenced to the machine zero point.
- You can move the workpiece zero point for all principal axes.

The workpiece zero point is a setup parameter.

Specify the workpiece datum

For more than one slide: Define the desired slide

Move the tool into position

Select Setting up > Shift zero point

The "displace zero point" dialog box displays the new workpiece zero point.

Touch the tool to the face

Contact position = tool zero point

Z=0

Accept the tool contact position as workpiece zero point

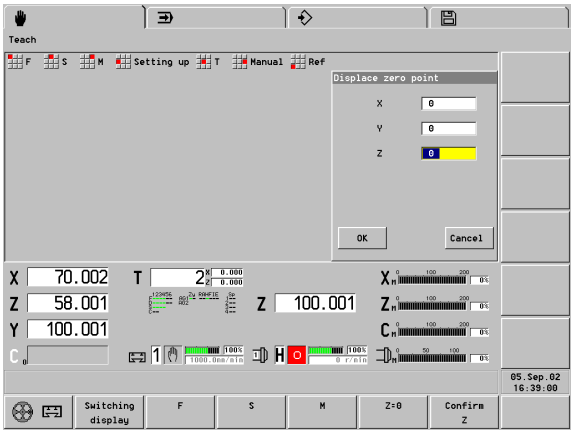
Workpiece zero point relative to the contact position

Confirm Z

Accept the tool contact position

Enter the measured value (distance of the tool contact position from the workpiece zero point)

Enter the position of the workpiece zero point

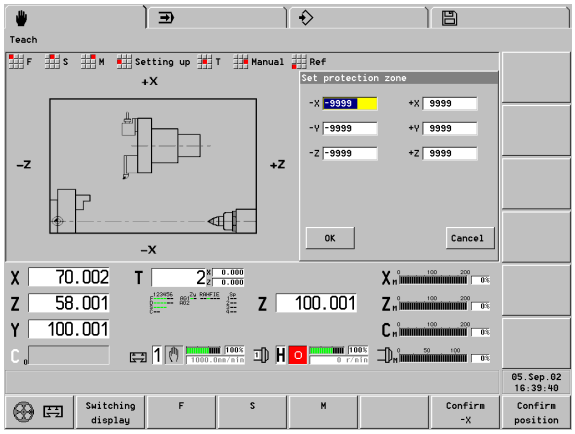


Defining the protection zone

Protection zone parameters:

- Serve for protection zone monitoring—not as software limit switches
- Given with respect to the machine zero point
- X values are radius dimensions
- 99999/-99999 means no monitoring of this protection zone side

The protection zone parameters are managed in the parameters MP 1116, 1156 and following.



Defining the protection zone

Insert any tool (except T0).

Select Setting up > Selection zones

Teach in the protection zone parameters per axis

Select the input field.

Position the tool to the protection zone limit.

Confirm -X — Accept the position as “protection zone -X” (or +X, -Y, +Y, -Z, +Z)

Teach in positive or negative protection zone parameters

Select any positive or negative input field.

Position the tool to the positive or negative “protection zone.”

Confirm position — Accept all positive or negative axis positions

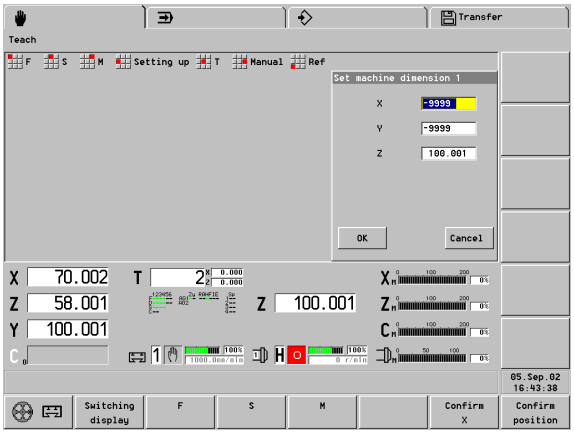
Enter the protection zone parameters

Setting up machine dimensions

This function takes into account the machine dimensions 1 to 9 and the “configured axes” per dimension. You can use the machine dimensions in the NC program.

Machine dimensions are managed in MP 7.

Machine dimensions are given with respect to the machine zero point.



Define machine dimensions

Select Setting up > Machine dimensions

Enter the “machine dimension number”

Teach in a single machine dimension

Select the input field.

Move the axis to the desired “position.”

Confirm X — Confirm the axis position as machine dimension (or Y or Z position).

Teach in all machine dimensions

Move the slide to the desired “position.”

Confirm position — Accept the axis positions of the slides as machine dimensions.

Enter machine dimensions

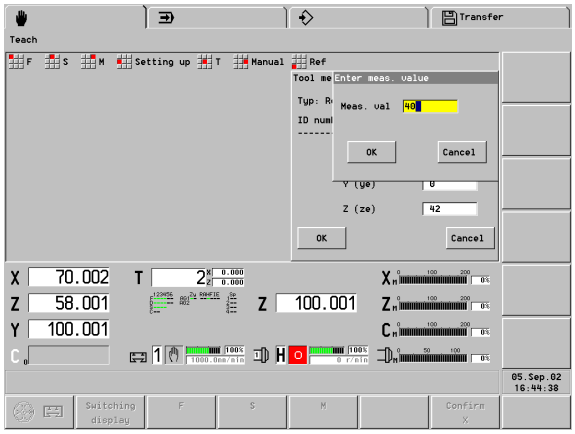
Enter the values (“Set machine dimension x” dialog box)

Tool measurement

You specify the type of tool measurement in MP 6:

- 0: Touch-off
- 1: Measurement with probes
- 2: Measurement using optical measuring systems

You move according to the measurement method to a certain position in the working space that is known to the system. Using that dimension, the CNC PILOT calculates the setting dimensions of the tool.



- The entries in the “Enter measured value” dialog box are given with respect to the workpiece zero point.
- The existing compensation values of the tool are deleted.
- The CNC PILOT enters the calculated tool dimensions into the database.

Tool measurement

Move the tool into position

Select Setting up > Tool set-up > Tool measuring.

The “Tool measuring T...” dialog box indicates the current tool dimensions.

Finding the tool dimensions by touch-off with the tool

Select the input field X; touch-off the diameter.

Confirm X	Load the diameter

Select the input field Z; touch-off the face.

Confirm Z	Accept Z position

Tool measurement with a touch probe

Select the X/Z input field

Move the tool tip in X/Z direction to the touch probe. The CNC PILOT loads the “dimension X/Z.”

Measure tools with an optical gauge

Select the X/Z input field

Align the tool point in the X/Z direction with the cross hairs.

Confirm X

Load the value (or Z position)

Enter the tool dimensions

Calculate the tool compensation

Move the tool into position

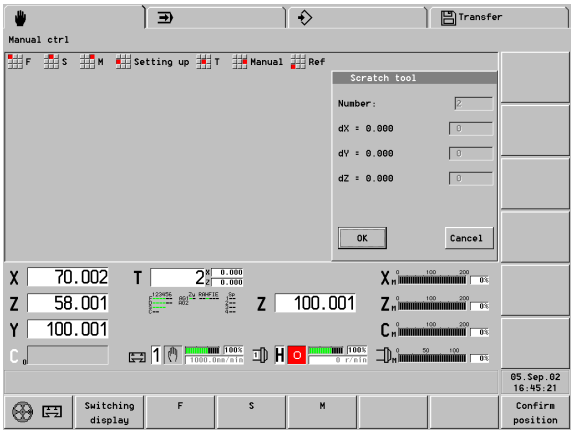
Select Setting up > Tool set-up > Tool compensation

Assign the handwheel to the X axis and move the tool by the compensation value

Assign the handwheel to the Z axis and move the tool by the compensation value

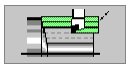
Confirm position

The CNC PILOT loads the compensation values.



3.5 Automatic Mode

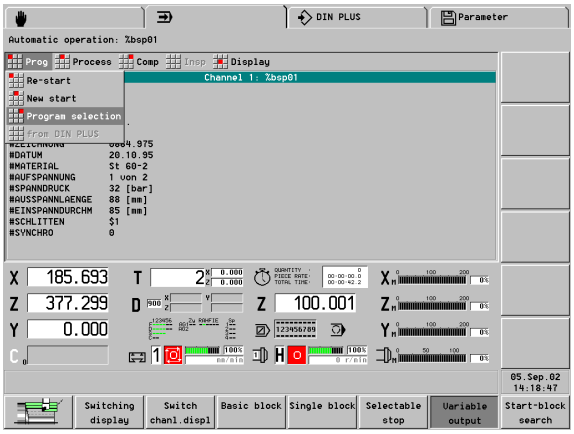
In Automatic mode, the data are entered and displayed according to control parameter 1 either in **meters** or in **inches.** The setting in the “program head” of the NC program governs the execution of the NC part program. It has no influence on operation or display.

Overview of soft keys in Automatic mode	
	Switch to “graphic display”
Switching display	Switch the machine display
Switch chanl.displ	Define block display for more channels
Basic block	Display basic blocks (individual paths of traverse)
Variable output	Suppress/permit variable output
Single block	Set single-block mode
Selectable stop	Program stop at M01 (selectable stop)
Start-block search	Run a start-block search

Program selection

The CNC PILOT interprets the NC program before it can be activated with Cycle Start. "#-Variables" are entered during the translation process. A "restart" prevents a new translation, while a "new start" forces a new translation.

- If the "turret table" of the NC program is not the currently valid table, there is a warning.
- The name of the NC program is retained until you select another program, even if the lathe was switched off in the meantime.



Program selection

Select Prog > Program selection The CNC PILOT opens the list of NC programs.

Select the NC program

The NC program is loaded without previous translation if:

No changes were made to the program or tool list.

The lathe was not switched off since the program was last selected.

Restart

Select Prog > Restart selection

The most recently active NC program is loaded without previous translation if:

No changes were made to the program or tool list.

The lathe was not switched off since the program was last selected.

Start again

Select “Prog > New start”

The NC program is loaded and **converted.**

(Use for starting an NC program containing # variables.)

From DIN PLUS

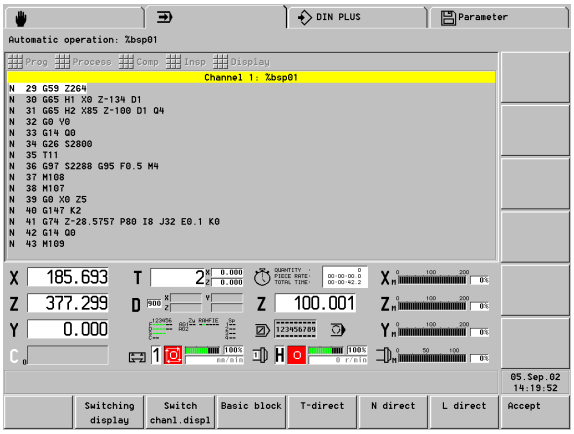
Select “Prog > From DIN PLUS”

The NC program selected in DIN PLUS is loaded and **converted.**

Finding a start block

In the start block search:

- Starting at program start, the CNC PILOT complies with the technology commands but does not conduct a tool change.
- The CNC PILOT allows no positioning.



Danger of collision

- If the start block includes a T command, the CNC PILOT first rotates the turret.
- The first traversing instruction is executed from the current tool position.
- Select a suitable start block on **all** slides before you press the Accept soft key.

Mid-program startup

Start-block search — Activating a start block search

Position the cursor on the start block. (The soft keys support your search.)

N direct — Enter the N number. The cursor is positioned to the block number

T-direct — Enter the T number: The cursor is positioned on the next T command

L direct — Enter the L number: The cursor is positioned to the next subprogram call

Accept — The CNC PILOT begins with the start-block search

Starts with the selected NC block

Modifying the program run

Skip level

The NC blocks with skip levels are not executed when the skip level is active. The “Skip levels” display field marks the (active) skip levels detected by the “block execution.”



The CNC PILOT accounts for activated/deactivated skip levels after approx. 10 blocks (reason: block scan during the execution of NC blocks).

Activating/deactivating a skip level

Select “Process > Skip level.”

Activating a skip level

Enter the “Level no.” and two or more skip levels as a “sequence of digits.”

Deactivating a skip level

No entry in “Level no.”

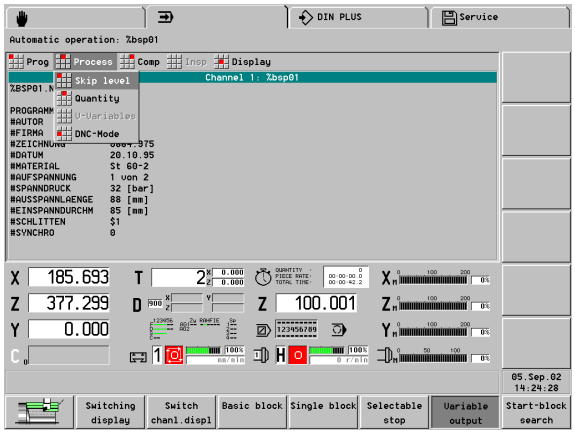
Production with defined quantity

Select “Process > Quantity”

Enter the quantity

Working with defined quantities:

- Counting range: 0..9999
- The quantity is counted after every program run.
- When an NC program is activated with “Program selection,” the CNC PILOT resets the counter.
- When a program has completed the quantity of parts, you can no longer start the NC program. To start the program again, press “Restart.”
- The Quantity remains stored even after the lathe is switched off.
- Quantity=0: No limit. The counter counts up the parts.
- Quantity > 0: The CNC PILOT manufactures the defined quantity; the counter counts down the parts.



“Skip levels” display field



Meaning:

- Upper row: Entered skip levels
- Lower row: Active skip levels

V variables

Working with V variables:

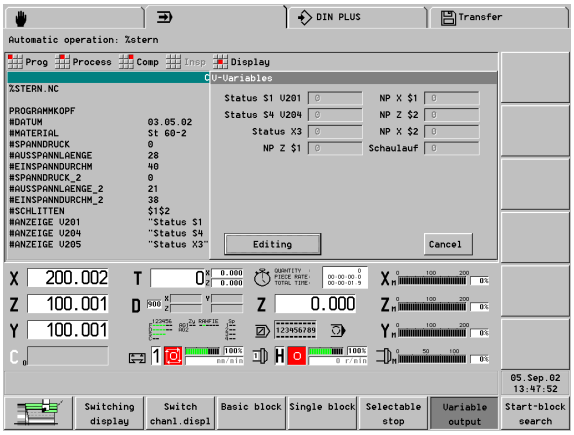
- The “V variables” dialog box serves for input and display of variables.
- V variables are defined at the beginning of the NC program. The meaning is specified in the NC program.

To check or enter a V variable:

Select “Process > V variables”

The CNC PILOT displays the defined variables in the NC program.

Change the variable: Press the Edit button



Single-block mode

In single-block mode, only one NC command (basic block) is executed at a time. The CNC PILOT then stops the cycle.

Set single-block mode

Single block	Activate "Single step" mode.
	Cycle Start runs the next NC command

Selectable stop

If the selectable stop is active, the CNC PILOT stops with **M01** and goes into the feed stop condition.

Program run with selectable stop

Selectable stop	Activate selectable stop

The CNC PILOT goes with an M01 into the feed stop condition.

	Cycle start resumes the program run

Status of selectable stop	
Selectable stop off	
Selectable stop on	

Feed rate override F%

The programmed feed rate changes with the programmed feed rate (range of 0 % to 150 %). The machine display shows the current feed rate override.

Adjusting the feed rate override

Adjust the desired override with the override knob (in the machine operating panel)

Spindle speed override

With the spindle speed override you can deviate from the programmed feed rate (range of 50 % to 150 %). The machine display shows the current spindle speed override.

Adjusting the spindle speed override



Spindle speed at 100% (of the programmed value)



Increase speed by 5%



Reduce speed by 5%

Compensation

Tool compensation

- Select “Comp > Tool compensation”
- **T number:** The CNC PILOT shows the active T number and the compensation values. You can enter a different T number.
- Enter the compensation values.
- Values entered here are added to the existing compensation values.



Tool compensation values:

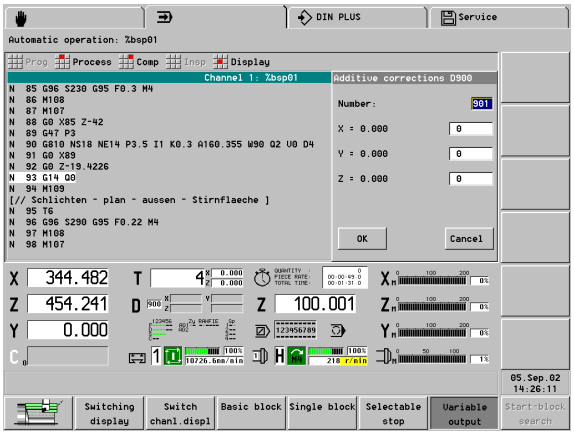
- Effective beginning with the next positioning command
- Transferred into the database
- Can be changed by no more than 1 mm

Additive compensation

- Select “Comp > Tool compensation values”
- Enter the compensation values (901 to 916). The CNC PILOT displays the valid compensation values.
- Enter the compensation values.
- Values entered here are added to the existing compensation values.

Additive compensation values:

- Activated with G149
- Managed in the set-up parameter 10
- Can be changed by no more than 1 mm



Tool life management

During Automatic mode, in the “tool life management,” switch off and on the ready-for-use status of a tool, or update the tool life data.

Changing the tool life data

Select Comp > Tool life management.

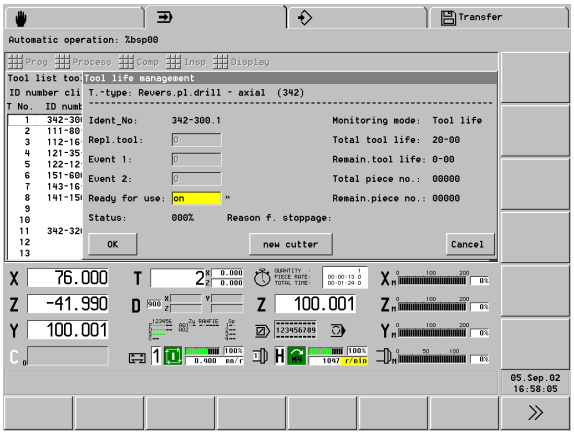
The CNC PILOT displays the tool list with the current tool life data.

Select the tool location.

Press ENTER. The CNC PILOT opens the “Tool life management” dialog box.

Select “Ready for use”

Press the “new cutter” button to update the tool life data.



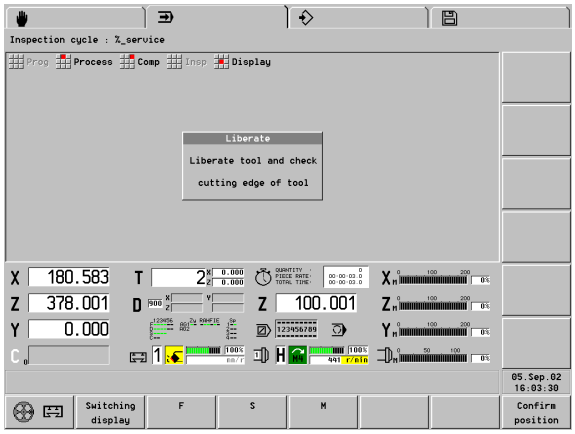
Inspection mode

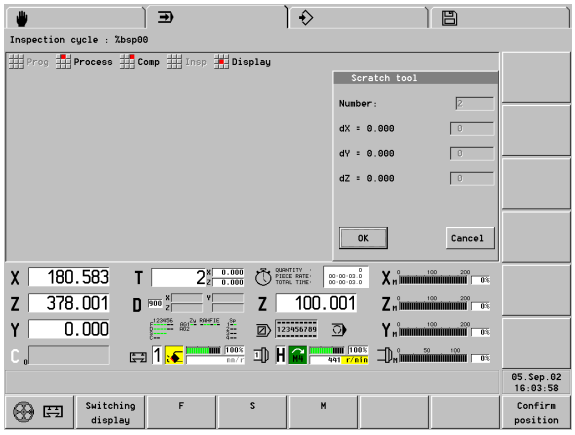
For the inspection mode, interrupt the program run, check or correct the active tool, or change the cutter. Resume the NC program at the point of interruption.

When the tool is retracted, the CNC PILOT stores the first five traverse movements. Each change in direction corresponds to a path of traverse.

Notes on the inspection mode:

- During the inspection process you can turn the turret, press the spindle keys, etc. The return motion program inserts the “correct” tool.
- When changing the cutting edge, select the compensation values so that the tool stops before the point of interruption.
- In the cycle stop condition you can interrupt the inspection cycle with the ESC key and switch to “Manual control.”





The inspection cycle is executed as follows:

1. Interrupt the program sequence and retract the tool.
2. Check the tool and replace the cutting edge if necessary.
3. Retract tool

1. Inspection – Retract the tool

Interrupt the program run

„Select INSPection

To retract the tool, use the manual direction keys.

If necessary, swivel the turret.

2. Inspection – Check the cutting edge

Inspect the tool; if necessary, replace it.

Confirm position	Conclude the inspection process. The CNC PILOT loads the return motion program (_SERVICE).

The "Tool compensation" dialog box appears. Enter the tool compensation.

If you are using a **new cutting edge,** modify the tool compensation so that the tool—when returning—comes to a stop **before** the point of interruption.

If necessary, activate the spindle.

3. Inspection – Return the tool to the work

At the beginning of the returning program, the control asks the two questions "Scrambled takeoff on restart?" and "Start from/before interruption point." Your answers determine the return motion program as follows:

- Scramble takeoff = yes (see 3.1 Return the tool and "scrambled takeoff")
  - Move **to** point of interruption: The return motion program positions the tool at rapid traverse to the interruption point and continues the program without stopping.
  - Move to **before** point of interruption: The return motion program positions the tool in rapid traverse to a position before the interruption point and continues the program without stopping.
- Scramble takeoff = no (see 3.2 Return the tool and stop)
  - Move **to** point of interruption: The return motion program positions the tool to the interruption point and stops the program.
  - Move to **before** point of interruption: The return motion program moves the tool to a position before the interruption point and stops the program.

"Scrambled takeoff = yes" is usually used when the cutting edge is not exchanged.

3.1 Return the tool and “scrambled takeoff”



Start the return motion program.

The “Scrambled takeoff on restart?” dialog box appears. Enter 1 (=yes)

Takeoff from IP:

The “Start from point of interruption (IP)” dialog box appears. Enter 0 (= to IP)

The return motion program positions the tool from/before the interruption point and continues the program **without stopping.**

Takeoff before IP:

The “Start from point of interruption (IP)” dialog box appears. Enter 1 (=before IP)

Then, in the “Distance from the interruption point” dialog box, enter the distance from the point of interruption

The return motion program positions the tool **before** the interruption point and continues the program **without stopping.**

The inspection cycle has been completed.

3.2 Return the tool and stop



Start the return motion program.

The “Scrambled takeoff on restart?” dialog box appears. Enter 0 (=no)

Takeoff from IP:

The “Start from point of interruption (IP)” dialog box appears. Enter 0 (=from IP)

The return motion program positions the tool on/before the interruption point and **stops.**

Takeoff before IP:

The “Start from point of interruption (IP)” dialog box appears. Enter 1 (=before IP)

Then, in the “Distance from the interruption point” dialog box, enter the distance from the point of interruption

The return motion program positions the tool before the interruption point and **stops.**



Resume the program run. The inspection cycle has been completed.

Select “Insp(ection)” again

The “Scratch with tool” dialog box opens (for information)

Assign the handwheel to the X/Z axis and “scratch” the workpiece.

Press “Accept value” to save the compensation values defined per handwheel.



Resume the program run. The inspection cycle has been completed.

If the NC program stops **before** the interruption point, the "Distance from the interruption point" comes into play:

- If the entered distance is greater than the distance between the start of the NC block and the interruption point, program sequence begins at the start of the interrupted NC block.
- If the entered distance is less than the distance between the start of the NC block and the interruption point, the CNC PILOT takes the distance into account.

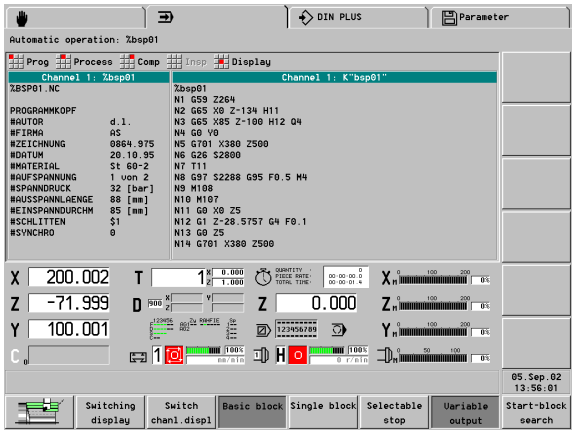
Block display, variable output

The CNC PILOT differentiates between:

- **Block display:** Lists the NC blocks according to the programmed sequence.
- **Basic block display:** The cycles are "resolved." Individual paths of traverse are shown. The numbering of the basic blocks is independent of the programmed block numbers.

Activate the basic block display:

Basic block	▶ Switch on/off the basic block display



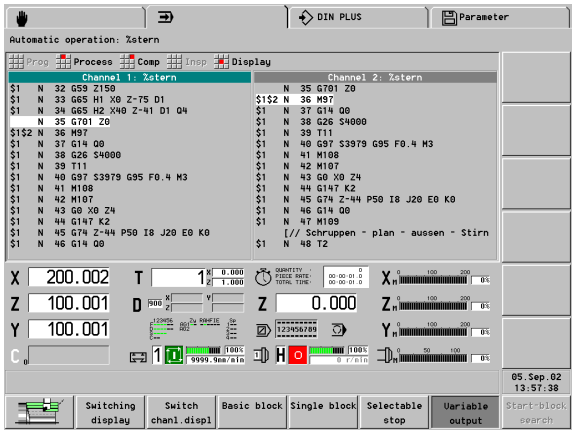
Channel display

For lathes with several slides, activate the block display for up to 3 channels.

Switching the channel display:

Switch chanl.displ	▶ Each time the soft key is pressed it connects a channel. Then only one channel is displayed.

If the block display is active for one channel, the basic block is shown in the right window. If the block display is active for two or more channels, the basic block display replaces the block display.



Font size

The font size of the block display can be adjusted in the menu.

- ▶ "Display > Font size > Smaller" reduces the font size
- ▶ "Display > Font size > Larger" increases the font size

Variable output

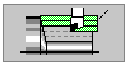
Variable output	▶ Pressing the soft key enables the variable soft key output (with PRINTA). Otherwise the variable output is suppressed.

Load monitoring display: see "Load Monitoring" on page 100

Graphic display

The “Automatic graphics” function displays the programmed blank and finished part and the paths of traverse. This enables process control of non-visible areas during production and provides an oversight of production status, etc.

All machining operations, including milling, are depicted in the turning window (XZ view).



▶ Activate the graphics. If the graphic was already active, the screen is adapted to the current machining status.

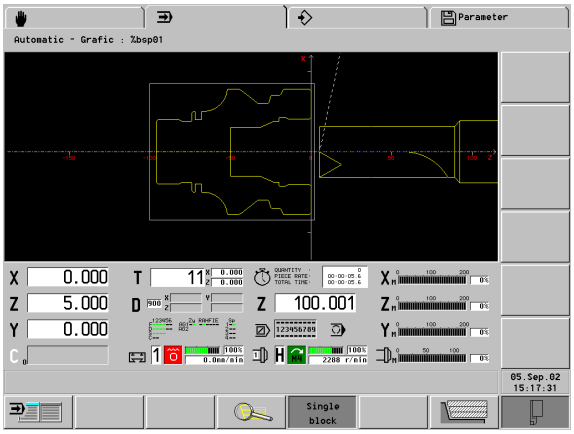


▶ Return to block display

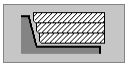
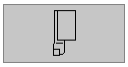
With the soft keys shown in the table you can influence the depiction of traverse.

In the standard setting, the CNC PILOT draws the complete path of traverse after every block. In the “motion” setting, the cutting process is shown synchronously to the production process.

- If no blank part was programmed, the standard blank form (control parameter 23) is assumed.
- “Motion” must be stopped at the beginning of the NC program. For program section repeats (M99), motion start in the next program run.



“Graphic display” soft keys

Single block	Set single-block mode
	Display of the paths of traverse (see “Path display” on page 367): ■ Line, or ■ Cutting trace
	Tool depiction (see “Screen layout, soft keys” on page 363): ■ Point of light, or ■ Tool
Motion	Displaying the cutting synchronously to the production process

Enlargement, reduction, and selecting a section

Zoom settings by keyboard:



- Activate the magnify function The red rectangle indicates the new section.
- Adjust the section:
  - To magnify, use the PgDn key
  - To reduce, use the PgUp key
  - To move the frame, use the arrow keys

ESC

- Exit the zoom function. The new section is displayed

Zoom settings by touchpad:

- Position the cursor to one corner of the section
- While holding the left mouse key, pull the cursor to the opposite corner of the section
- Right mouse key: Return to standard size

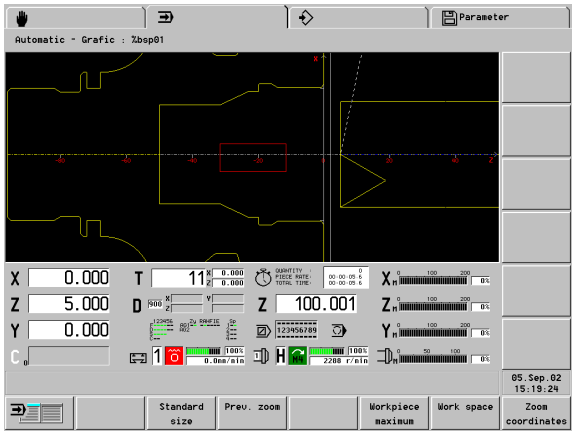
ESC

- Exit the zoom function. The new section is displayed.



After having enlarged a detail to a great extent, select “Workpiece maximum” or “Work space,” and then isolate a new detail.

You can make the standard setting by soft key (see table). The “By coordinates” setting (simulation window and position of the tool zero point) is given with respect to the selected slides.

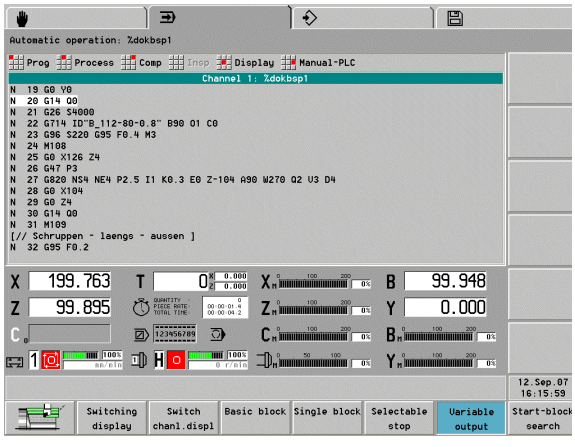


“Graphic display” soft keys	
Standard size	Last setting was “workpiece maximum” or “working space”
Prev. zoom	Cancels the last magnification
Workpiece maximum	Depicts workpiece in the largest possible magnification
Work space	Depicts the work space including the tool change position
By coordinates	Set simulation window and position of the workpiece zero point

Mechatronic tailstock

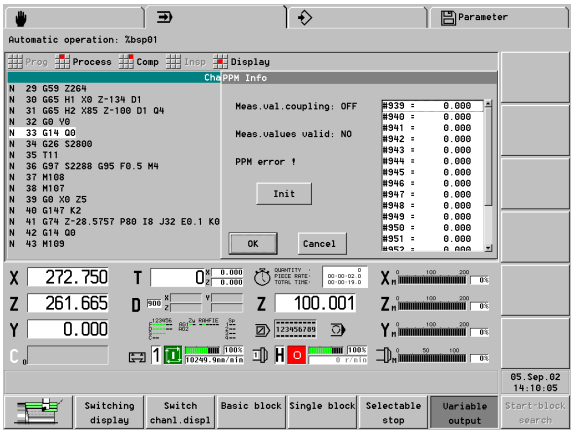
A moveable opposing spindle can be used as a mechatronic tailstock if the machine tool builder prepares the machine for this function.

If so, start the sleeve mode with the “Manual PLC” menu item. A prerequisite is that the Automatic mode was stopped with cycle stop or that an M0/M01 in the NC program triggered a cycle stop.



Post-process measuring status

Post-process measuring means that the workpiece is measured outside the lathe and that the results are transferred to the CNC PILOT. The “PPM Info” dialog box contains information on the status of the measured values, displays the transferred results, and allows initialization of communication with the measuring device.



Operation of “post-process measuring”:

- Select Display > PPM Status
- The PPM Info dialog box provides information on the status of the measured values and the most recently transferred results.
- If you select “Init,” the post-process measuring function is re-initialized and all measured values are deleted.

PPM Info dialog box:

- Measured-value coupling (same as control parameter 10)
  - Off: Measuring results are immediately transferred. Previous values are overwritten.
  - On: Measuring results are not transferred until the previous results have been processed.
- Measured values valid: Status of the measured values (after the measured values have been transferred with G915, the status “No” is displayed).
- #939: Total result of last measurement
- #940..956: Measuring results last transferred by the measuring function

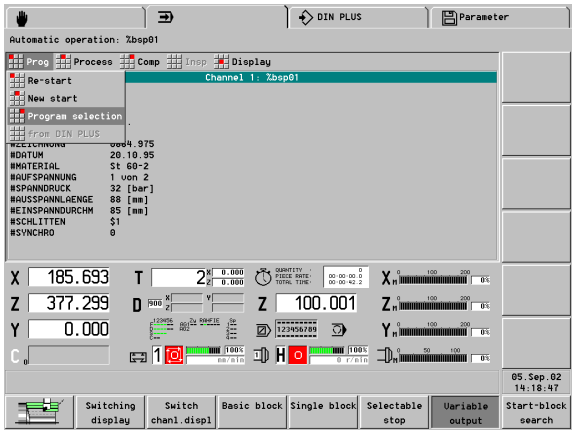


The post-process measuring function saves the measured values received in the clipboard. The PPM Info dialog box displays in #939..956 the values contained in the clipboard—not the variables.

3.6 Machine Display

Switching the display

The machine display of the CNC PILOT can be configured. Per slide, you can configure up to 6 displays in Manual mode and Automatic mode (starting with control parameter 301).



Switching the display

Switching display

▶ Switch to the next configured display.

▶ Switch to the display of the next slide.

▶ Switch to the display of the next spindle.

Position display

In “Display type” (MP 17) you set the values of the position display:

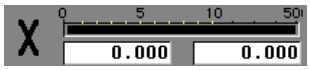
- 0: Actual values
- 1: Following error
- 2: Distance of traverse
- 3: Tool tip with respect to machine zero point
- 4: Slide position
- 5: Distance from reference dog to zero pulse
- 6: Nominal position
- 7: Difference from tool point to slide position
- 8: Nominal director position

Display elements

The following table explains the standard display fields. For further display fields: see “Control parameters for machine display” on page 580

Display elements	
	**Position display** (distance between tool tip and workpiece zero point) ■ Empty field: Axis has not been homed ■ Letter designating the axis appears in white: Axis disabled ■ Display values shown in gray (only with X or Z): The actual value display is **invalid** because the B axis was tilted.
	**Display of C position** ■ “Index”: marks the C axis “0/1” ■ Empty box: C axis is not active ■ Letter designating the axis appears in white: Axis disabled

Display elements



Distance-to-go display (distance remaining in the current traverse command)

- Bar graphic: Distance remaining in millimeters
- Field below left: Actual position
- Field below right: Distance to go



T display without tool life monitoring

- T number of active tool
- Tool compensation values



T display with tool life monitoring

- T number of active tool
- Tool life data



Quantity of workpieces/time per workpiece

- Number of finished workpieces in this batch
- Production time or current workpiece
- Total production time for this batch



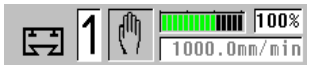
Load display

- Load of the spindle motors/axis drives with regard to rated torque



D display (additive compensation)

- Number of the active compensation
- Compensation values



Slide display

- Symbol appears in white: Disabled
- Numeral: Selected slide
  - White background: No “converting and mirroring” active (G30)
  - Color background: “Converting and mirroring” active (G30)
- Cycle status
- Bar graphic: Feed rate override in %
- Upper field: Feed rate override
- Lower field:
  - Current feed rate
  - With stationary slide: Nominal feed rate (gray text)
- Slide number on a blue background means the rear face is active

Cycle status:



Cycle on



Feed stop



Cycle off



Manual control

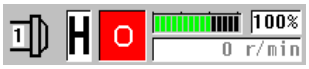


Inspection cycle



Set-up mode

Display elements

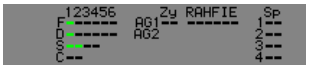


Spindle display

- Symbol appears in white: Disabled
- Numeral in spindle symbol: Gear range
- "H"/Numeral: Selected spindle
- Spindle status
- Bar graphic: Feed rate override in %
- Upper field: Spindle speed override
- Lower field:
  - Current speed
  - With stationary spindle: Nominal speed (gray text)
  - With position control(M19): Spindle position

Spindle status:

	Direction of spindle rotation M3
	Direction of spindle rotation M4
	Spindle stop
	Spindle in position control (M19)
	C axis is "activated"



Overview of enabled elements

Shows the enabling status of up to 6 NC channels, 4 spindles, and 2 C-axes. Enabled elements are marked (green).

- Display group at left: Enabled
  - F: Feed rate
  - D: Data
  - S: Spindle
  - C: C axis
  - 1..6: Number of the slide/spindle of the C axis
- Display group in center: Status
  - Zy – left dash: Cycle on/off
  - Zy – right dash: Feed Stop
  - R=Reference run
  - A: Automatic mode
  - H: Manual control
  - F: Retracting (after crossing the limit switches)
  - I: Inspection mode
  - E: Set-up switch
- Display group at right: Spindle
  - Display for direction of rotation left/right
  - Both active: Spindle positioning (M19)

3.7 Load Monitoring

The load monitoring function of the CNC PILOT compares the current torque, or the values for the work of the drives, with the values from a reference run.

If "torque limit 1" or the "work limit" is exceeded, the CNC PILOT marks the tool as worn out. If torque limit 2 is exceeded, the CNC PILOT assumes tool breakage and stops the machining process (feed stop). Violations of limit values are reported as error messages.

The load monitoring identifies worn tool in the "tool diagnosis bits." If you are using the **tool life monitoring** function, the CNC PILOT will manage the replacement of tools. As an alternative, you can evaluate the tool diagnosis bits in the NC program.

The load monitoring function defines the **monitoring zones** and the drives to be monitored (G995) in the NC program. The torque limits of a monitoring zone depend on the maximum torque determined by the reference machining cycle.

The CNC PILOT checks the values for torque and work in each interpolator cycle and displays the values in a time reference grid of 20 ms. The limit values are calculated from the reference values and the limit factor (control parameter 8). You can later change the limit values in "Edit load parameters."



- Make sure that the conditions for reference machining comply with those for production (feed-rate/speed override, tool quality, etc.).
- The CNC PILOT monitors up to four components per monitoring zone.
- Using "G996 Type of load monitoring," you can control the hiding the rapid traverses paths and the monitoring of torque and/or work.
- The graphic and numeric displays are relative to the rated torque values.

Machining using load monitoring

During use of the load monitor, a worn tool should require significantly more torque than a sharp one. As a rule, drives that are subjected to considerable loads should be monitored. That usually means the spindle.

Due to the relatively small torque variations, it is difficult to monitor machining operations with small cutting depths.

A decrease in torque cannot be identified.

Defining the **monitoring zones:** The reference values for torque depend on the maximum torque values of the zone. Therefore, lower torque values cannot always be monitored.

Facing with constant cutting speed: The spindle is monitored as long as the acceleration is <= 15 % of the mean value from maximum acceleration and maximum braking deceleration (MP 811, ...). Since acceleration increases as a result of the increase in rotational speed, the CNC PILOT usually only monitors the period after the first cut.

Empirical values for the machining of steel:

- For longitudinal turning the cutting depth should be > 1 mm.
- For recessing the cutting depth should be > 1 mm.
- For hole drilling, ensure that the diameter drilled is between 6 and 10 mm.

Reference machining

The reference machining cycle (registration of nominal values) determines the maximum permissible torque and work of each monitoring zone. These values are used as **reference values**!

CNC PILOT executes a reference machining cycle if:

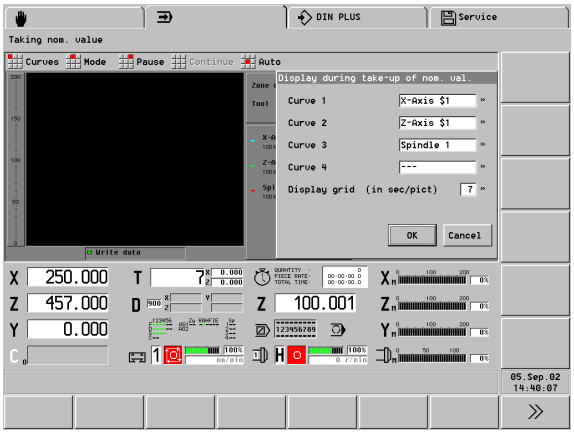
- There are no monitoring parameters
- You select “yes” in the reference machining dialog box (after program selection).

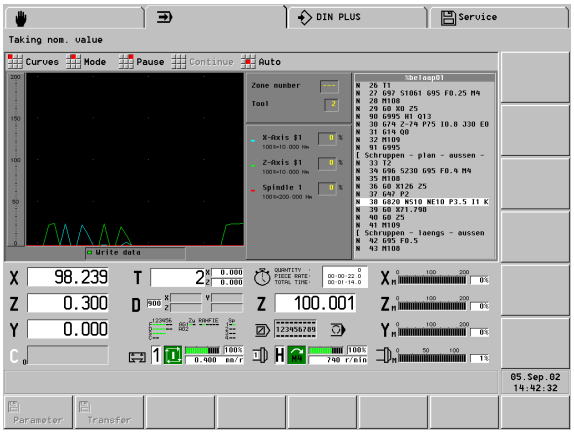
Activate the display:

▶ „Disp(lay) > select “load monitoring display”: the CNC PILOT switches to the “Taking nominal values” submenu

“Taking nominal values” submenu:

- **“Curves” menu item**
  - In curves 1 to 4 you assign the input field to the drives.
  - With “Display grid” you influence the accuracy of the depiction. A small value increases the accuracy (values: 4, 9, 19, 39 seconds per image).
- **“Mode” menu group**
  - **Line graphic:** Shows the torque over the time axis.
  - **Bar graphic:** Shows the torque as a bar and indicates the peak values.
  - **Save/do not save measured values:** The measured values must be saved for later analysis of reference machining. Check the setting in the “Save data” (“Write data”) display field.
  - **Overwrite/do not overwrite limit values:** Setting of whether the limit values are overwritten when reference machining is run again.
- **Pause:** Stops the display
- **Continue:** Resumes the display
- **Auto:** Return to Automatic menu



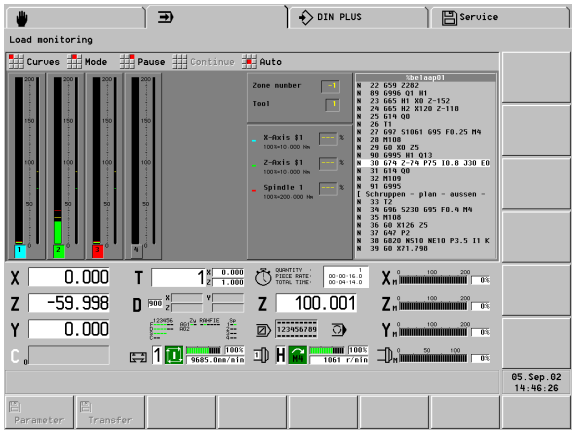


Additional information during recording:

- Zone number: Present monitoring zone
- Negative algebraic sign: Production is not monitored (example: skipping the paths of rapid traverse).
- Tool: Active tool
- Selected drives are listed and the current torque values are displayed.
- Block display

Production using load monitoring

If you wish to use the load monitoring function for your machining processes, you must activate it in the NC program (G996).



Display torque values and limit values:

▶ Select Disp(lay) > Load monitoring > Display

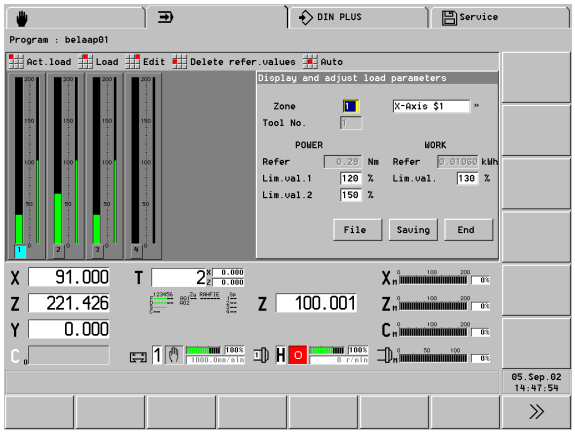
“Load monitoring > Display” submenu:

- **“Curves” menu item**
  In curves 1 to 4 you assign the input field to the drives.
  - Line graphic: A curve
  - Bar graphic: Up to four bars
  - With “Display grid” you influence the accuracy of the depiction. A small value increases the accuracy (values: 4, 9, 19, 39 seconds per image).
- **“Mode” menu group**
  - **Line graphic:** Shows the torque over the time axis and the limit values. Limit values gray: Nonmonitored area (hiding rapid traverse paths).
  - **Bar graphics:** Displays the current torque, the values for work and all limit values of the monitoring zone.
- **Pause:** Stops the display
- **Continue:** Resumes the display
- **Auto:** Return to Automatic menu

Editing limit values

The function for editing the load parameters allows you to analyze reference machining cycles and optimize limit values.

The CNC PILOT displays the program name of the loaded monitoring parameters in the header.



Selection:

▶ Select Disp(lay) > Load monitoring > Edit

“Load parameter editor” submenu:

- **Act. load** (Load current file) menu item: Monitoring parameters of the active NC program.
- **Load:** Monitoring parameters you have selected.
- **Edit:** Display and edit the limit values.
- **Delete reference values:** Delete the load parameters of the NC program.
- **Auto:** Return to Automatic menu

Editing the load parameters

The “Display and adjust load parameters” dialog box displays the parameters of **one** component **of one** monitoring zone, which can then be edited.

The bar graphic shows all components of the monitoring zone (the larger bar displays the values for performance; the smaller bar displays the values for work). The selected component is highlighted.

Enter the monitoring zone and select the component. The CNC PILOT displays the reference values. The limit values for performance and work, which are displayed, can be edited. The tool (T number) is displayed for information.

Buttons of the dialog box:

- **Saving:** Save the limit values of the component in the specified zone.
- **End (or ESC key):** Exit the dialog box.
- **File:** Switch to “Line graphics.”
  Precondition: The values measured during the reference machining cycle have been stored.

Analyzing reference machining

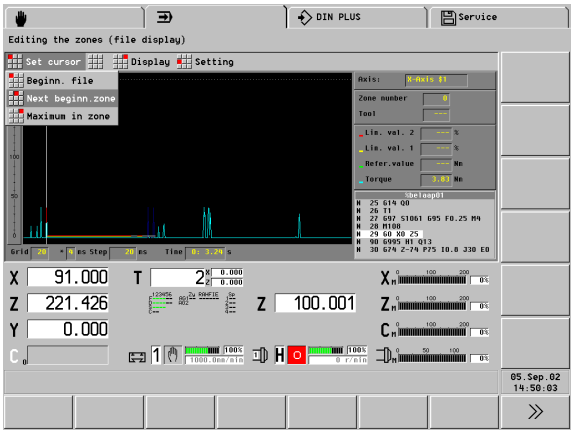
The load monitor shows the torque and the limit values of the selected component over time.

- Limit values gray: Nonmonitored area (hiding rapid traverse paths).
- In addition the CNC PILOT displays the values of the cursor position numerically.

Selection:

▶ Selection: File button in the “Display and adjust load parameters” dialog box.

ESC ▶ Switch back to “Edit load parameters”



“Analyzer (file display)” submenu:

- **Set cursor:** Position the cursor with the left/right arrow keys or to the
  - **Beginning of file**
  - **Next beginning zone**
  - **Maximum in zone**
- **Display** menu item: Select the component from the “Display file” dialog box.
- **Setting - Zoom**: Set the zoom factor. (Small values increase the accuracy of the display and reduce the step size of the cursor.)

  The settings defined for the grid, the time reference grid of the measured value registration and the cursor position are shown in the line below the graphic display relative to the start of reference machining. Time 0:00.00 sec = Start of reference machining cycle.

Load monitoring parameters

Machine parameters for load monitoring (spindle: MP 809, 859, ...; C axis: MP 1010, 1060; linear axes: MP 1110, 1160, ...):

- **Start-up time for monitoring** [0..1000 ms] is evaluated if “Omit paths of rapid traverse” is active:
  - **Spindles:** A limit value is calculated from the acceleration and brake ramps. As long as nominal acceleration exceeds the limit value, the monitoring function is deactivated. As soon as nominal acceleration falls below the limit value, the monitoring function is delayed by the start-up time for monitoring.
  - **Linear and C axes:** After rapid traverse has changed to feed rate, the monitoring function is delayed by the “start-up time for monitoring.”
- **Number of sample values to be averaged [1..50]**
  The mean value reduces the sensitivity to short-term peak load during monitoring.
- **Maximum torque** of drive [Nmm]
- **Delay in reaction P1, P2 [0..1000 ms]:** The CNC PILOT indicates that torque limit 1/2 has been exceeded after the time “P1/P2” has passed.

Control parameter 8 “Load monitoring settings”

- **Factor for torque limit value 1, 2**
- **Factor for work limit value**
  Limit value = reference value * factor for limit value
- **Minimum torque [% of rated torque]:** Reference values below this value are raised to this minimum torque value. This prevents that limit values are exceeded as a result of minor torque differences.
- **Maximum file size [KB]:** If the data during measured value registration exceed the maximum file size, the “oldest” values are overwritten. Approximate value: 12 KB for one component per minute of program-run time.

Control parameter 15 “bit codes for load monitoring”:
Assigns the bits number used in G995 to the drives (logical axes).



4

DIN Programming

4.1 DIN Programming

Introduction

The CNC PILOT supports both conventional DIN programming and DIN PLUS programming.

- **Conventional DIN/ISO programming:** You program the basic contour with line segments, circular arcs and simple turning cycles. For conventional DIN programming, the simple tool description is sufficient.
- **"DIN PLUS" (ISO) programming:** The geometrical description of the workpiece and the machining process are separated. You first program the geometry of the blank and finished part. Then you machine the workpiece, using contour-related turning cycles. The **contour follow-up** function can be activated for each machining step, including individual paths of traverse and simple turning cycles. The CNC PILOT optimizes the machining process as well as the paths for approach and departure (no noncutting passes).

Depending on the type and complexity of your machining task, you can use either simple DIN programming or DIN PLUS programming.

NC program sections: The CNC PILOT supports the division of the NC program into individual program sections.

- Program head (organizational data and setup information)
- Tool list (turret table)
- Chucking equipment table
- Workpiece-blank definition
- Description of machined part
- Machining of workpiece

Parallel operation: While you are editing and testing programs, your machine can execute **another** NC program.

Example: Structured DIN PLUS program

```
PROGRAMMKOPF [PROGRAM HEAD]
#MATERIAL [MATERIAL] St 60-2
#EINSPANNDURCHM          120 [CLAMP. DIAM.]
#AUSSPANNLAENGE          106 [CLAMP. LENGTH]
#SPANNDRUCK 20 [CLAMP. PRESSURE]
#SCHLITTEN                          $1 [SLIDE]
#SYNCHRO                                    0

REVOLVER 1 [TURRET]
T1 ID"342-300.1"
T2  ID"111-80-080.1"
. . .
SPANNMITTEL [CHUCKING EQUIPMENT] (Zero
point shift Z282)
H1 ID"KH250"
H2 ID"KBA250-77" Q4.

ROHTEIL [BLANK]
N1 G20 X120 Z120 K2

FERTIGTEIL [FINISHED PART]
N2 G0 X60 Z-115
N3 G1 Z-105
. . .
BEARBEITUNG [MACHINING]
N22 G59 Z282
N23 G65 H1 X0 Z-152
N24 G65 H2 X120 Z-118
N25 G14 Q0
[Predrilling 30 mm outside centric face]
N26 T1
N27 G97 S1061 G95 F0.25 M4
. . .
ENDE [END]
```

DIN PLUS screen

Screen layout:

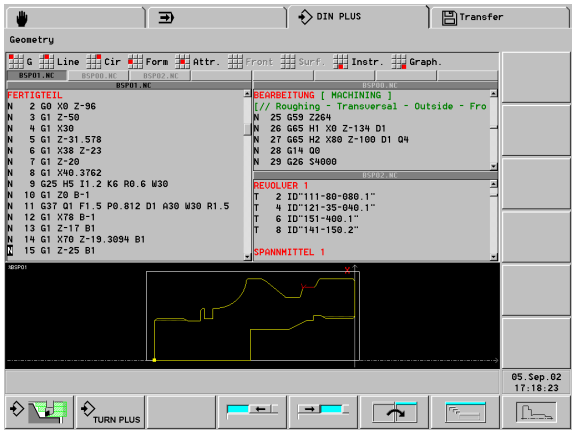
1. Menu bar
2. NC program bar with the name of the loaded NC programs. The selected program is marked.
3. Full, double or triple editing window. The selected window is marked.
4. Contour display or machine display
5. Soft keys

Parallel editing: You can edit up to eight NC program/subprograms in parallel. The CNC PILOT displays NC programs as desired in either a full, double, or triple window.

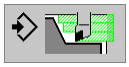
Main menus and submenus: The functions of the DIN PLUS editor are contained in the main menu and various submenus. The submenus can be called by

- selecting the desired menu item
- positioning the cursor in the respective program section

Soft keys: Soft keys are available for fast switching to "neighboring operating modes" for changing the editing window and for activating the graphics.

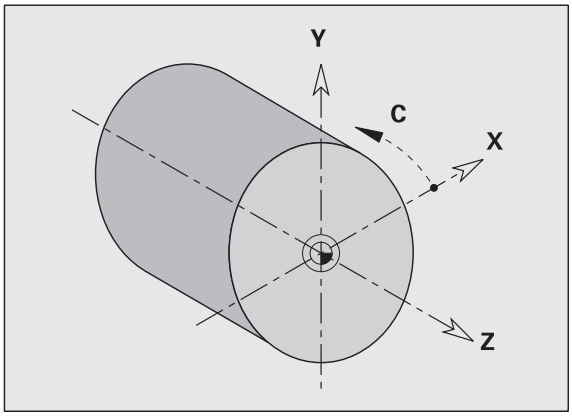


Soft keys

	Change to the simulation mode
	Change to the TURN PLUS mode
	Switch the NC program
	Switch the NC program
	Switch the editing window
	Select full-size window (one editing window)
	Select double or triple window
	Activate the graphics

Linear and rotary axes

Principal axes: Coordinates of the X, Y and Z axes refer to the workpiece zero point.

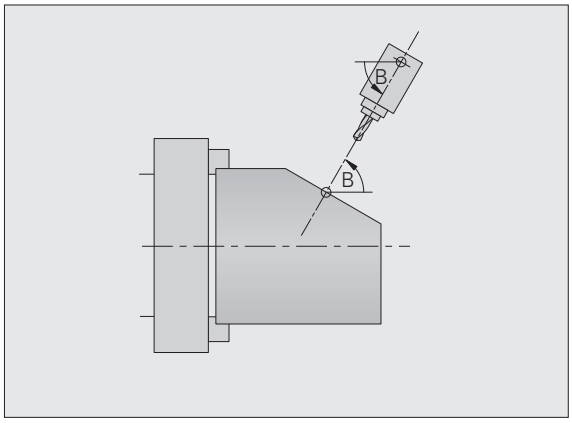


Remember about **negative X coordinates**

- Not allowed with contour descriptions
- Not allowed for turning cycles
- The contour follow-up is switched off
- The direction of rotation for circular arcs (G2/G3, G12/G13) has to be set manually
- The position of the tool radius compensation (G41/G42) has to be set manually

C axis as reference axis:

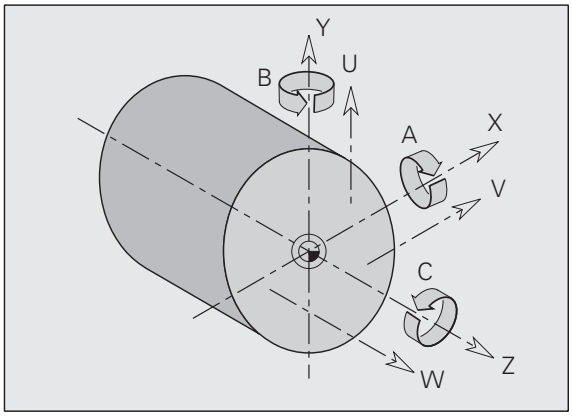
- Angle data are with given respect to the zero point of the C axis.
- C-axis contours and C-axis operations:
  - Positions on the front/rear face are entered in Cartesian coordinates (XK, YK), or polar coordinates (X, C)
  - Positions on the lateral surface are entered in polar coordinates (Z, C). Instead of C, the **linear value CY** is used ("unrolled" reference diameter).



B axis—tilted working plane: The B axis makes it possible to drill, bore and mill in oblique planes. The coordinate system is tilted for programming in such a way that you can define the drilling patterns and milling contours in the YZ plane. Machining is then conducted in the tilted plane.

Secondary axes (auxiliary axes): In addition to the principal axes, the CNC PILOT supports:

- U:Linear axis in X direction
- V:Linear axis in Y direction
- W:Linear axis in Z direction
- A:Rotary axis that rotates about X
- B:Rotary axis that rotates about Y
- C:Rotary axis that rotates about Z



The auxiliary axes are only programmed in the MACHINING section, using the functions G0 to G3, G12, G13, G30, G62 and G701. A circular interpolation is only possible in the principal axes. Rotary axes (auxiliary axes) are programmed in the MACHINING section, using G15.

- The DIN editor respects only address letters of the configured axes.
- The behavior of the **rotary axes B and C** depends on whether they have been configured as principal or auxiliary axes.

Units of measurement

You write NC programs in metric or inch values. The unit of measure is defined in the "Unit" box (see "PROGRAM HEAD section" on page 136).



Once the unit of measure has been defined, it cannot be edited any longer.

Elements of a DIN program

A DIN program consists of the following elements:

- Program number
- Program section codes
- NC blocks
- Commands for program structuring
- Comment blocks

The **program number** begins with "%" followed by up to 8 characters (numbers, uppercase letters or underscore; no diacritical marks) and the extension "nc" for main programs or "ncs" for subprograms. The first character must be a number or a letter.

Program section codes: When you create a new DIN program, certain program section codes are already entered. You can add new codes or delete existing ones, depending on your program requirements. A DIN program must contain at least the MACHINING and END section codes.

NC blocks begin with an N followed by a block number (with up to four digits). The block numbers do not affect the sequence in which the program blocks are executed. They are only intended for identifying the individual NC blocks.

The NC blocks of the PROGRAM HEAD, TURRET and CHUCKING EQUIPMENT sections are not included in the block number organization of the DIN editor.

An NC block contains **NC commands** such as positioning, switching or organizational commands. Traversing and switching commands begin with G or M followed by a number (G1, G2, G81, M3, M30, ...) and the address parameters. Organizational commands consist of key words (WHILE, RETURN, etc.), or of a combination of letters/ numbers.

You can also program NC blocks containing only variable calculations.

You can program several NC commands in one NC block, provided they have different address letters and do not have opposing functions.

Examples:

- Permissible combination: N10 G1 X100 Z2 M8
- Non-permissible combination:
  N10 G1 X100 Z2 G2 X100 Z2 R30 (same address letters are used more than once) or
  N10 M3 M4—opposing functionality

NC address parameters
The address parameters consist of 1 or 2 letters followed by

- A value
- A mathematical expression
- A question mark (VGP simplified geometry programming)
- A letter "i" to designate incremental address parameters (examples: Xi..., Ci..., XKi..., YKi..., etc.)
- A **# variable** (calculated during NC program interpretation)
- A **V variable** (calculated at run time)

Examples:

- X20(Absolute dimension)
- Zi–35.675(Incremental dimension)
- X?(Simplified geometry programming)
- X#12(Variable programming)
- X{V12+1}(Variable programming)
- X(37+2)*SIN(30)(Mathematical expression)

Program branches and repeats

- You can use program jumps, repeats and subprograms to structure a program. Example: Machining the beginning/end of a bar etc.
- **Skip level:** Influences the execution of individual NC blocks
- **Slide code:** You can assign the NC blocks to a specific slide provided that your lathe is equipped with more than one slide.

Input and output: With "input" the machine operator can influence the flow of the NC program. Using "output," you can communicate with the machinist. Example: The machinist is required to check measuring points and update compensation values.

Comments are enclosed in brackets "[...]." They are located at the end of an NC block or in a separate NC block.

4.2 Programming Notes

Configuring the DIN editor

The following properties of the DIN editor can be configured in the main menu:

- Display/do not display a help graphic next to the dialog box
- Number of editing windows
- Font size

You save and load these settings.

Help graphic:

- Select Config > Help graphic. The editor opens the “Configuration of help graphic” dialog box.
- Set whether the help graphic should be displayed.

Number of editing windows:

- Select Config > Window > Full-size window (or > Double window, > Triple window). The editor sets the selected number of windows.

Font size:

- Select Config > Font size > Less than (or > Greater than). The editor reduces/enlarges the font size.
- Select Config > Font size > Adjust fonts The editor uses the font size of the selected window for all editing windows.

Saving/loading editor settings:

- Select Config > Settings > Save. The editor saves the editor settings.
- Select Config > Settings > Load. The editor loads the saved editor settings including the NC program.
- Select Config > Settings > Auto-save on. The editor saves the condition during switch-off.
- Select Config > Settings > Auto-save off. The editor condition is not saved during switch-off.

Parallel editing

The CNC PILOT runs up to eight NC program/subprograms in parallel and provides up to three editing windows.

Editing window: Set full-size or multiple windows:

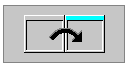


▶ Set full-size window



▶ Set multiple windows (whether two or three windows is set in the configuration)

Switch the editing window:



▶ Press the soft key, or

▶ use the touchpad to click the desired window

Change the NC program:



▶ Press the soft key



▶ Press the soft key, or

▶ use the touchpad to click the NC program in the NC program bar.

Selecting submenus, positioning the cursor

The submenus can be called by

▶ selecting the desired menu item

▶ positioning the cursor in the respective program section



▶ Press the ESC key to return to the main menu.

When you call the menu items **Geo**metry,**Pro**cessing, "Turret assignment" or "Chucking equipment," the CNC PILOT jumps to the corresponding program section.

When you position the cursor in the BLANK, FINISHED PART or MACHINING section, the CNC PILOT switches to the corresponding submenu.

Position the cursor:

▶ "Prog > Prog(ram) start" positions to the beginning of the program

▶ "Prog > Prog(ram) end" positions to the end of the program

▶ With the cursor keys or PgUp or PgDn

Making, editing and deleting NC blocks

Make NC block:

The insertion of new NC blocks varies depending on the program section.

Program head

- Close the "Editing program head" dialog box: The CNC PILOT automatically creates the blocks of the program head (code "#").

TURRET and CHUCKING EQUIPMENT program sections:

Ins

- Press the INS key: The CNC PILOT opens the dialog for a new tool or chuck.
- The new block is inserted after the dialog is finished.

Contour programming, programming the machining process and programming in subprograms:

Ins

- Press the INS key: The CNC PILOT makes a new block below the cursor position.
- Alternatively you can program the NC command directly. The CNC PILOT makes a new NC block or inserts the NC command in the existing NC block.

Delete the NC block:

- Position the cursor on the NC block to be deleted.

DEL

- Press the DEL key: The CNC PILOT deletes the NC block.

Add an NC element:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.).
- Insert NC element (G, M, T function, etc.).

Change NC element:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.) or the section code.
- Press ENTER or double-click with the left mouse key. The CNC PILOT activates a dialog box which displays the block number, the number of the G or M function, or the address parameters of the function, which can then be edited.

When editing section codes, you can change the associated parameters (Example: number of the turret). When you edit NC words (G, M, T), the CNC PILOT activates a dialog box for editing the address parameters.

Delete NC elements:

- Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.) or the section code.



- Press the DEL key. The element highlighted by the cursor **and** all the related elements are deleted. (Example: If the cursor is located on a G command, the address parameters are also deleted.)



Before deleting a complete NC block, the CNC PILOT displays a confirmation request. Individual elements of an NC block including G or M functions are deleted by the editor without confirmation prompt.

Search functions

The search function of the DIN editor supports:

Block no. search:

- Select “Block > Search functions> Search block” in the main menu. The editor opens the “Block no. search” dialog box.
- Enter the block number and close the dialog block: The CNC PILOT positions the cursor on the block number (if it exists).

NC word search (G command, address parameter, etc.):

- Select “Block > Search functions> Search word” in the main menu. The editor opens the “Word search” dialog box.
- As of software version 625 952-02: Press <CTRL+F>. The editor opens the “Word search” dialog box. To continue searching, press <F>.
- Enter the NC word and close the dialog box. The cursor skips to the first NC block containing the word. The CNC PILOT searches from the cursor position to the end of the program, then continues searching from the start of the program.

Conversational or free editing

In **free editing** you select the NC functions from the menus and edit the address parameters in dialog boxes.

In **free editing** you enter all elements of the NC block. The maximum block length for "free editing," is 128 characters per line.

To select "free" editing:

- Select "Block > New: Free input" in the main menu. The DIN editor inserts an NC block at the cursor position and waits for the entry of a complete NC block.
- Select "Block > Change: Free input" in the main menu. The DIN editor presents the NC block at the cursor position for editing.

Geometry and machining commands

The **G commands** are divided into:

- **Geometry commands** for describing the blank and finished part.
- **Machining commands** for the MACHINING section.



Some G functions are used for blank/finished-part definition and in the MACHINING section. When copying or shifting NC blocks, keep in mind that "geometry" functions are used exclusively for describing a contour, while "machining" functions are used only in the MACHINING section.

Contour programming

The "contour follow-up" function and contour-related turning cycles require the previous description of the blank and finished part. For milling and drilling, contour definition is a precondition if you wish to use fixed cycles.

Contours for turning

- Describe a continuous contour.
- The direction of the contour description is independent of the direction of machining.
- CNC PILOT closes open contours paraxially.
- Contour descriptions must not extend beyond the turning center.
- The contour of the finished part must lie within the contour of the blank part.
- When machining bars, define only the required section as blank.
- Contour definitions apply to the entire NC program, even if the workpiece is rechucked for machining the rear face.
- In the fixed cycles, the defined contour is used to program "reference values."

To describe **blank parts,** use

- G20 "Blank part macro" for standard parts (cylinder, hollow cylinder).
- G21 "Cast-part macro" for blank-part contours based on finished-part contours.
- Individual contour elements (such as are used for finished-part contours) where use of G20 or G21 is not possible.

To describe finished parts, use individual contour elements. The contour elements or the complete contour can be assigned attributes accounted for during the machining of the workpiece (example: roughness, oversizes, etc.).

For intermediate machining steps, define **auxiliary contours.** Auxiliary contours are programmed in the same way as finished-part descriptions. One contour description is possible per AUXILIARY CONTOUR. You can make multiple AUXILIARY CONTOURS.

Contours for C-axis machining:

- Contours for C-axis machining are programmed within the FINISHED PART section.
- Identify the contours as a FRONT or SURFACE. You can use section codes more than once or program multiple contours within one section code.

Multiple contours in an NC program

The CNC PILOT supports up to four contours (workpiece blank and finished part) per NC program. The CONTOUR section code introduces the description. Parameters on zero point shift and the coordinate system define the position of the contour in the working space. A G99 in the machining section assigns the machining to a contour.

Contour Follow-Up

The CNC PILOT takes the blank part as a basis and accounts for each cut and each cycle when regenerating the contour. Thus you can inspect the current contour of the workpiece during each machining stage. With the "contour follow-up" function, the CNC PILOT optimizes the paths for approach and departure and avoids noncutting passes.

Contour regeneration is available only for turning operations. It also works with auxiliary contours.

Preconditions for contour follow-up:

- Workpiece-blank definition
- Proper description of tools ("simple definition of tools" is not sufficient)

Block references

When editing G commands related to the contour (MACHINING section), activate the contour simulation and load the block references from the displayed contour.



- Place the cursor in the input box.
- Switch to the contour display
- Place the cursor on the desired contour element
- Press ENTER to load the block number of this contour element

Contour generation during simulation

You can save contours generated in the simulation and transfer them into the NC program. Example: You describe the blank form and finished part and simulate machining of the first setup. Then you save the contour. You define a shift of the workpiece zero point and/or a mirror image. The simulation saves the "generated contour" as the workpiece blank and the originally defined finished part contour, taking the zero point shift and mirroring into account.

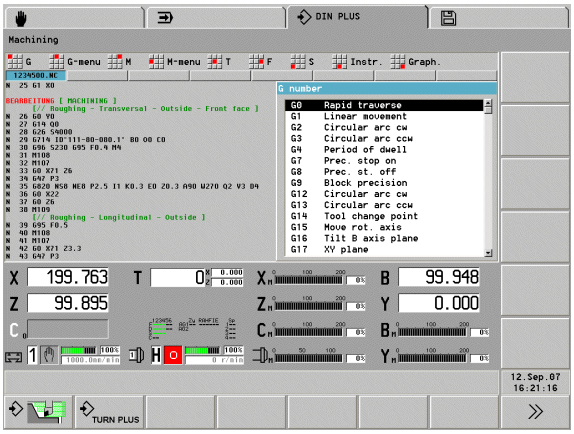
Download the contours generated for the blank and finished part:

- Position the cursor.
- Select "Block > Insert contour" in the main menu.

List of G functions

If you do not know the G number, the DIN editor supports you with the G function list.

- Select G in the geometry or machining menu. The editor opens the G number list.
- Place the cursor on the desired G function.
- Press ENTER to load the G number.



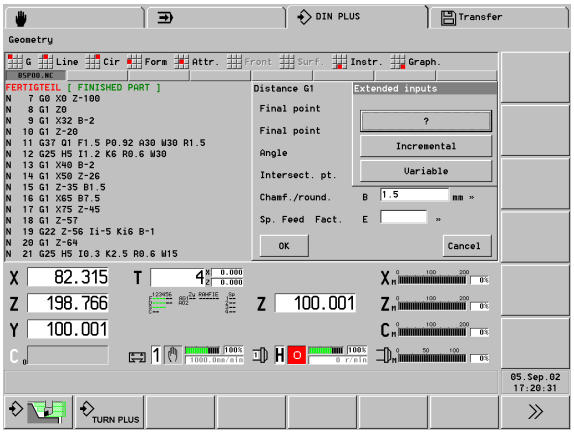
Address parameters

Coordinates can be programmed absolutely or incrementally. If you do not make any entry for X, Y, Z, XK, YK, C, the coordinates of the block previously executed will be retained (modal).

The CNC PILOT calculates missing coordinates in the principal axes X, Y or Z if you program "?" (simplified geometry programming).

The machining functions G0, G1, G2, G3, G12 and G13 are modal. This means that the CNC PILOT uses the previous G command if the address parameters X, Y, Z, I or K in the following block have been programmed without a G function However, the address parameters must have been programmed as absolute values.

The CNC PILOT supports the use of variables and mathematical expressions as address parameters.



To edit address parameters:

- Call the dialog box.
- Position the cursor in the input field and enter/change the values, or



- „Call "extended input"
  - „?" programming (simplified programming)
  - Switch from incremental to absolute, or vice versa
  - Activate variable input

Tool programming

The designations of the tool pockets are fixed by the machine tool builder. Each tool holder has a unique **T number.**

In the T command (MACHINING section) you program the tool holder, and therefore the position to which the tool carrier rotates. The CNC PILOT retrieves the assignment of the tools to the tilted position from the TURRET section, or the tool list (in case the T number is not defined in TURRET).

Multipoint tools: A multipoint tool is a tool with more than one cutting edge. During T call, the T number is followed by an S to identify the cutting edge.

T number.S (S=0 to 4)

S=0 identifies the main cutting edge, which does not need to be programmed. In the TURRET section, you define only the main cutting edge.

When a cutting edge of a multipoint tool has become dull, the tool life management function marks all cutting edges as worn-out.

Examples:

- T3 or T3.0: Tilted position 3; main cutting edge
- T12.2: Tilted position 12; cutting edge 2

Replacement tools: If you use the **tool life monitoring,** you define a sequence of exchange. As soon as a tool is worn out, the CNC PILOT inserts a replacement tool. The CNC PILOT does not stop the program run until the last tool of the tool sequence of exchange is worn out.

In the TURRET section and the T commands, you program the first tool to be changed. The CNC PILOT automatically inserts the replacement tool. When programming variables (access to tool compensation or tool diagnosis bits), you also address the first tool of the sequence. The CNC PILOT automatically addresses the active tool.

You can define replacement tools in "Setup."

Subprograms, expert programs

Subprograms are used to program the contour or the machining process.

In the subprograms, transfer parameters are available as variables. You can fix the designation of the transfer parameter (see “SUBPROGRAM section” on page 145).

In every subprogram, the variables #256 to #285 are available for internal calculations.

Subprograms can be nested up to six times. Nesting means that a subprogram calls a further subprogram, etc.

If a subprogram is to be run repeatedly, enter the number of times the subprogram is to be repeated in the Q parameter.

The CNC PILOT distinguishes between local and external subprograms.

- **Local subprograms** are in the file of the NC main program. Local subprograms can only be called in from their corresponding main programs.
- **External subprograms** are stored in separate NC files and can be called in from any NC main program or other NC subprograms.

Expert programs

An expert program is a subprogram that executes complex processes and is adapted to the machine configurations (for example, workpiece transfer to opposing spindles). Expert programs are usually provided by the machine tool builder.

NC program conversion

For variable programming and user communication, keep in mind that the CNC PILOT interprets the complete NC program before the program run.

The CNC PILOT differentiates between:

- **# variables** that are calculated during the conversion of the NC program
- **V variables** are calculated at run time, which means during the execution of an NC block.
- Inputs/outputs during **NC program conversion**
- Inputs/outputs during **NC program run**

Fixed cycles

HEIDENHAIN recommends programming a fixed cycle as follows:

- Tool change
- Define the cutting data
- Position the tool in front of the working area
- Definition of safety clearance
- Cycle call
- Retract the tool
- Move to tool change position



Danger of collision!

Remember when omitting cycle programming steps during optimization:

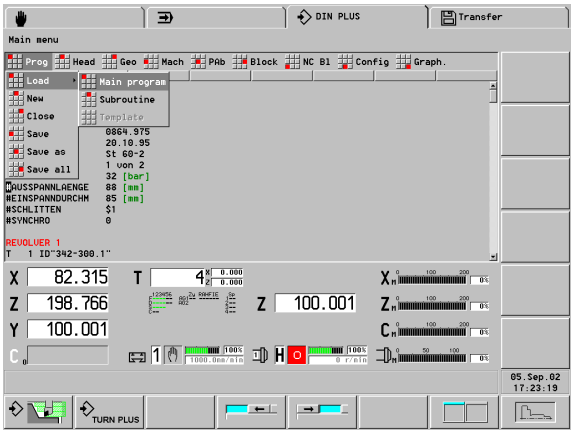
- A special feed rate remains in effect until the next feed command (for example the finishing feed rate during recessing cycles).
- Some cycles traverse diagonally back to the starting point if you use the standard programming (for example roughing cycles).

Typical structure of a fixed cycle

`. . .`	
`BEARBEITUNG [MACHINING]`	
`N.. G59 Z..`	Zero point shift
`N.. G26 S..`	Define the speed limit
`N.. G14 Q..`	Move to tool change position
`. . .`	
`N.. T..`	Insert the tool
`N.. G96 S.. G95 F.. M4`	Define the technology data
`N.. G0 X.. Z..`	Pre-position
`N.. G47 P..`	Definition of safety clearance
`N.. G810 NS.. NE..`	Cycle call
`N.. G0 X.. Z..`	If necessary, retract
`N.. G14 Q0`	Move to tool change position
`. . .`	

4.3 The DIN PLUS Editor

Overview of main menu



The **“Prog” pull-down menu** (program management) contains the following functions for NC main and subprograms:

- Load existing NC programs
- Create new NC programs
- Save new or edited NC programs

The **“Head” menu group** (NC program header) contains functions for editing

- the program header
- the turret assignment
- the chucking table

With the **“Geo” menu item** (geometry) you can branch to programming the workpiece blank or finished part contour. You select either a workpiece blank or position the cursor to the BLANK or FINISHED PART section and switch to the geometry menu.

The **“Mach” menu item** (machining) calls the submenu for programming the machining. At the same time the CNC PILOT positions the cursor to the MACHINING section.

The **“Pab” menu item** (program section code) calls a selection box with section codes. It enables you to insert additional codes into your NC program.

The **“Block” menu group** contains functions for marking and editing groups of NC program blocks.

The **NC Bl menu group** contains

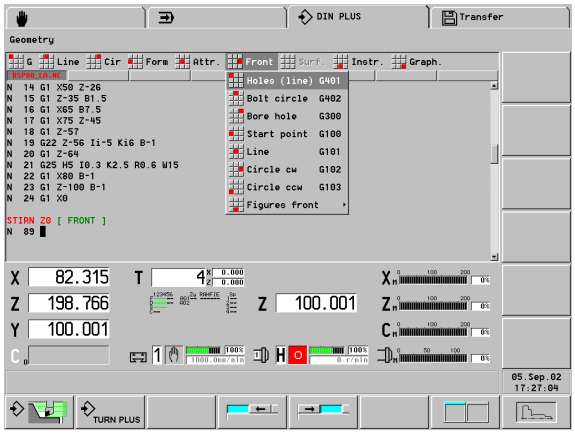
- Functions for cursor positioning
- Functions for numbering the NC blocks
- Search functions
- “Free editing” function

In the **“Config” menu group** configuration you can make the following settings:

- Switch-off/on the help graphic
- Window configuration
- Font size
- In addition you can manage the settings”

In the **“Graph” menu group** you set the graphic window and activate/deactivate the contour display.

Overview of geometry menu



The **Geometry** submenu contains G functions and instructions for the BLANK and FINISHED PART sections.

With the menu items **G, Line and Cir** you select basic contour elements:

- If you know the G number, call G and enter the number of the G function.
- If you do not know the G number, select the “Line” or the desired “Cir” (circle)

The **Form menu group** contains the following form elements:

- Recesses
- Undercuts
- Thread
- Centric hole
- It also provides subprogram calling

In the **Attr. menu group** (attribute) you define the following attributes, which are assigned to contours or contour sections:

- Precision stop
- Surface roughness
- Oversizes
- Special feed rates
- Additive compensation

The **Front menu group** contains figures, patterns and elements for defining milling contours for the front and rear faces. This menu item cannot be selected until the cursor is placed in the corresponding program section.

The **Surf. menu group** (lateral surface) contains figures, patterns and elements for defining milling contours for the lateral surface. This menu item cannot be selected until the cursor is placed in the corresponding program section.

The **Instr. menu group** (instructions) contains

- Section codes
- Instructions for structuring the program
- Programming with variables
- Comments

The **Graph. menu item** activates or updates the graphic window.

Overview of machining menu

The **Machining** submenu contains G, M, T, S and F functions as well as further instructions for the MACHINING section.

Selecting the **G and M functions:**

- If you know the G or M number, call the G or M menu and then enter the number of the function.
- If you do not know the G or M number, select the desired function from the G or M menu.

T menu items (tool call):

- Select T
- Enter the T number, or select the tool from the list

Menu item F:

- Select F. The editor calls G95—Feed per revolution.

Menu item S:

- Select S. The editor calls G96—Cutting speed

The **Instr. menu group** (instructions) contains

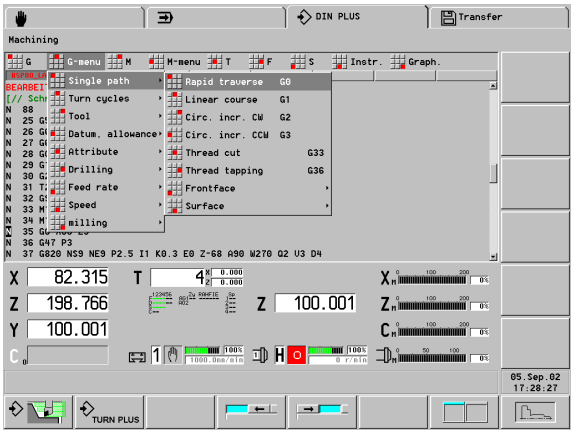
- Section codes
- Instructions for structuring the program
- Programming with variables
- Subprogram calls
- Comments
- Templates
- Working plan

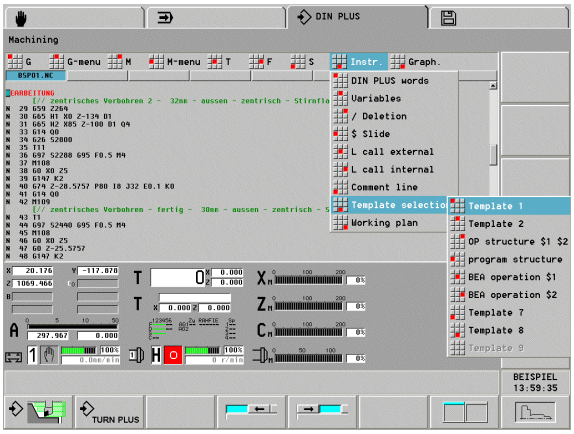
The **Graph. menu item** activates or updates the graphic window.

A **template** is a predefined NC code block that is adapted to your lathe and is integrated in the NC program. Templates generally contain structure commands, synchronizations, zero point shifts etc. They make it easier to program complex operations.

Templates are provided by the machine tool builder. He can inform you whether templates are available on your machine and if so, which ones. You can optimize the templates for your requirements (see “DIN PLUS Templates” on page 354).

The **working plan** function collects all comments that begin with // and places them before the MACHINING instruction. This gives you an overview of the NC program’s operations.





New NC program

NC programs contain instructions and information that are configured specially to your lathe and organization. You can combine these data in a starting template and use it to write new programs (see programming example). Such a sample program makes it easier to write a new program and aids in standardizing NC programs.

If you do not use the starting template, CNC PILOT makes a new NC program with the standard program section codes.

The degree of detail included in the starting template depends on the complexity of your machine, your organization and many other criteria.

For more on creating and editing your starting template: see "DIN PLUS Templates" on page 354.

To create a new NC program using a starting template:

- Select "Prog > New".
- Enter a program name.
- Make the settings of the NC main program.
- Press "OK." The CNC PILOT makes an NC program on the basis of the starting template (prerequisite: The DINSTART.bev file is in the NCPS directory)

To create a new NC program:

- Select "Prog > New"
- Enter a program name
- Make the settings of the NC main program
- Press the Program Head button: The NC editor creates the NC program and switches to the program head editing function.

To create a new subprogram:

- Select "Prog > New"
- Enter a program name
- Make the subprogram settings
- Press "OK." The NC editor creates the subprogram.

Example: "Starting template"

```
PROGRAMMKOPF [PROGRAM HEAD]
#MATERIAL                    St 60-2 [MATERIAL]
#MASCHINE [MACHINE]                   STANDARD
#SPANNDRUCK [CLAMP. PRESS.]                 40
#SCHLITTEN                          $1 [SLIDE]
#SYNCHRO                                     0

REVOLVER 1 [TURRET]

SPANNMITTEL [CHUCKING EQUIPMENT] (Zero
point shift Z...)
H1 ID"KH250"
H2 ID"KBA250-69" X 100 Q2

ROHTEIL [WORKPIECE BLANK]
N1 G20 X100 Z100 K2

FERTIGTEIL [FINISHED PART]
N2 G0 X0 Z0

BEARBEITUNG [MACHINING]
N22 G59 Z100 [enter zero point shift]
N23 G26 S4000 [enter speed limit]
N24 G65 H1 X0 Z-100 [enter chuck position]
N25 G65 H2 X100 Z-100
N26 G14 Q0


ENDE [END]
```

NC program management

Loading the NC program:

Load NC program in the next **free window:**

- Select Prog > Load > Main program (or > Subprogram). The CNC PILOT displays the files.
- Select and load the NC program or subprogram

Load NC program in **selected window:**

- Select and activate free editing window
- Select Prog > Load > Main program (or > Subprogram). The CNC PILOT displays the files.
- Select and load the NC program or subprogram

Saving the NC program:

To conclude the NC program editing:

- Select Prog > Close. If the NC program is new or changed, the CNC PILOT opens the Save NC Program dialog box
- Set whether and under what name the NC program is to be saved

Save the NC program of the active window:

- Select Prog > Save. The CNC PILOT saves the NC program but stays in the editing window.

Save the NC program of the active window under a new program name:

- Select Prog > Save As. The CNC PILOT opens the Save NC Program dialog box.
- Enter the file name and set whether the editing window should be closed

Save the NC programs of all active windows:

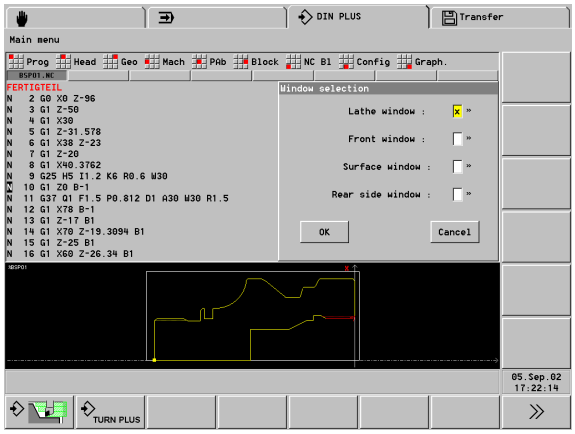
- Select Prog > Save All. The CNC PILOT saves all NC programs but they stay in the editing window.



- When you exit the DIN PLUS operating mode, the NC programs are saved automatically. The old version of the NC program is overwritten.
- If an NC program was changed but not saved, the program name is shown in red. If the NC programs were not changed or saved, the program name is shown in black.

Graphics window

During editing CNC PILOT displays programmed contours in up to two graphic windows.



To select the graphic windows:

- Select “Graph. > Window” in the main menu
- Mark the desired window

To activate the contour display/update the contour:

In the main menu:

- Select “Graph. > Graphic ON”



In the submenu:

- Press the soft key, or
- Select Graphic

To activate the machine display:

- Select “Graph. > Graphic OFF” in the main menu

Notes on the graphic window:

- The starting point of the turning contour is marked by a “small box.”
- When the cursor is located on a block of the “blank or finished part” section the corresponding contour element is highlighted in red and the direction of contour definition is indicated.
- When programming fixed cycles, you can use the displayed contour for establishing block references.
- The CNC PILOT starts at the base when displaying contours on the lateral surface (reference diameter for SURFACE).



- Additions/changes on the contours are not considered until GRAPHIC is selected again.
- Unambiguous NC block numbers are a prerequisite for the contour display!

Workpiece-blank programming

To describe the workpiece blank, proceed as follows:

Standard blank (cylinder, hollow cylinder):

- Select “Geo > Blank> Chuck piece/bar G20” in the main menu.
- The CNC PILOT
  - Creates an NC block in the BLANK section
  - Switches to the “Geometry” submenu
  - Activates the “Chuck part, cylinder/tube G20” dialog box

Casting as blank (the workpiece blank contour is based on the finished part contour):

- Select “Geo > Blank> Casting G21” in the main menu.
- The CNC PILOT
  - Creates an NC block in the BLANK section
  - Switches to the “Geometry” submenu
  - Activates the “Casting G21” dialog box

Any workpiece blank contour:

- Select “Geo > Blank> Free contour” in the main menu.
- The CNC PILOT
  - Positioning the cursor on the BLANK program section
  - Switches to the “Geometry” submenu
- Define through individual contour elements (like a finished part contour)

Block numbering

Setting the block numbering: The block numbering settings are the starting block number and the block-number increment. The first NC block receives the starting block number and the block-number increment is added for each further NC block. The settings for starting block number and block number increment are tied with the NC program. Call:

- Select “Block > Increment” in the main menu. The editor opens the “Config. of block no. increment” dialog box.
- Enter the starting block number and block number increment.

Renumbering the NC blocks: The editor

- renumbers the NC blocks.
- corrects the block references of contour-related G commands in the main program and all subprograms called in this main program.
- corrects the block references for subprogram calls.
- renumbers the NC blocks of a subprogram if this subprogram is used by the main program and is open in the editor.

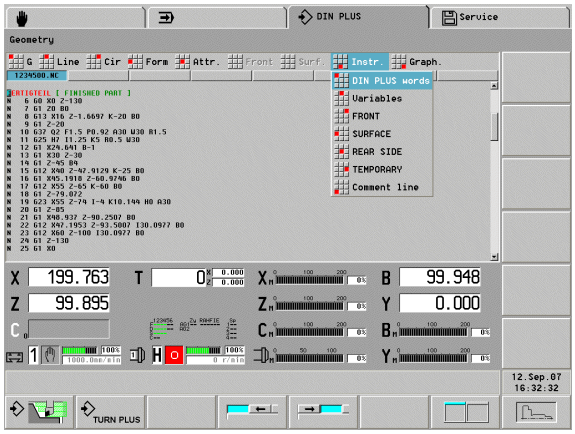
Call:

- Select “Block > Block numbering” in the main menu. The editor renumbers the NC blocks.

Programming instructions

Instructions of the geometry menu

Below are the contents of the Instr. (instructions) menu group



DIN PLUS words:

- Select “Instr. > DIN PLUS words.” The editor opens the selection box.
- Select the desired instruction for program structuring or the input/output command.

Variable:

- Select “Instr. > Variables.” The editor opens the input line.
- Enter a variable or mathematical expression.

Program section codes:

Auxiliary contour:

- Select “Instr. > AUXILIARY CONTOUR.” The editor enters the code in an appropriate position.

Front face, rear face or lateral surface contour:

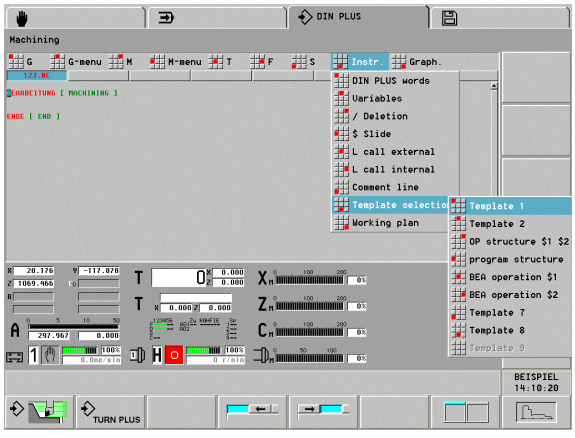
- Select „Instr.> FRONT FACE“ (or „.. > LATERAL SURFACE“, „.. > REAR FACE“)“. The editor opens the dialog for position input.
- Enter the position of the plane.

Comment:

- Select “Instr. > Comment line.” The editor opens the input line.
- Enter text. The comment is inserted above the position of the cursor.

Instructions of the machining menu

Below are the contents of the Instr. (instructions) menu group



DIN PLUS words:

- Select "Instr. > DIN PLUS words." The editor opens the selection box.
- Select the desired section code, instruction for program structuring or the input/output command.

Variable:

- Select "Instr. > Variables." The editor opens the input line.
- Enter a variable or mathematical expression.

Skip level:

- Select "Instr. > / Deletion." The editor opens the "Skip level" dialog box.
- Enter skip level [1 to 9].

Slide code:

- Select "Instr. > $ Slide." The editor opens the "Slide number" dialog box.
- Enter the slide number. Enter several slides as a sequence of numbers.

External subprogram call:

- Select "Instr. > L call external." The editor opens the selection list with the available subprograms.
- Select the desired subprogram and enter the transfer parameter.

Internal subprogram call:

- Select "Instr. > L call internal." The editor opens the "subprogram call" dialog box.
- Enter the subprogram name (block number with which the subprogram begins) and the transfer parameters.

Comment:

- Select "Instr. > Comment line." The editor opens the input line.
- Enter text. The comment is inserted above the position of the cursor.

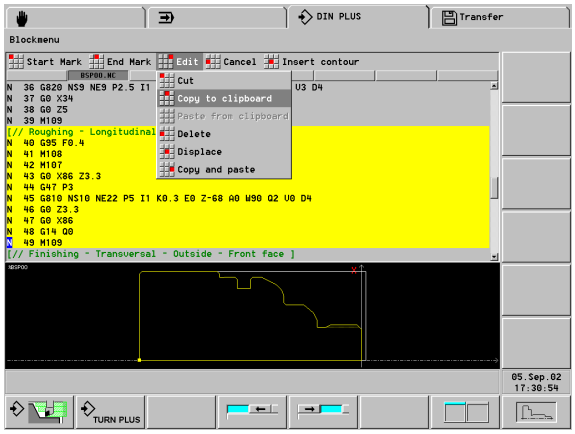
Templates:

- Select "Instr. > Template selection > Template selection ..."
  - The editor opens the dialog box of the template
  - After the dialog box is closed the template is taken into the NC program

Creating and NC program overview:

- Select "Instr. > Working plan."
- The editor
  - collects all comments that begin with "// ..."
  - places these comments in the MACHINING section

Block group menu



This function enables **NC block groups** (several successive NC blocks) to be moved, copied, deleted or exchanged between NC programs.

To define an NC block group, highlight the first and last line of the block group. Then select the desired operation on the block group.

In order to exchange blocks groups **between NC programs,** copy the block group to the clipboard. Then read in the block group from the clipboard. A block group remains in the clipboard until it is overwritten by another block group.

Marking a block group:

Beginning of block group:

- Position the cursor on the beginning of a block group.
- Press “Start mark”

Block group end:

- Position the cursor on the “end of a block.”
- Press “End mark”

Saving a block group in the clipboard:

Load the marked block group in the clipboard and delete it:

- Select “Edit >Cut.”

Copy the marked block group into the clipboard:

- Select “Edit > Copy to clipboard”

Get block group from clipboard:

- Position the cursor at the target position.
- Select “Edit > Paste from clipboard” The block group is inserted at the target position.

Deleting a block group:

- Select “Edit >Delete.” The editor deletes the marked block group definitively (it is not stored in the clipboard).

Moving a block group:

- Position the cursor at the target position.
- Select “Edit >Displace.” The marked block group is moved from its initial position to the position indicated by the cursor.

Copying a block group:

- Position the cursor at the target position.
- Select "Edit > Copy and paste." The block is inserted at/copied to the target position.

The "Cancel" menu item:

- Select "Cancel." The editor removes all marking.

The "Insert contour" menu item

- Select "Insert contour." The editor inserts the most recent workpiece blank and finished part contour in the simulation below the cursor position

As an alternative of the block group menu, you can use the usual **WINDOWS key combinations** for marking, deleting, shifting etc.:

- Marking by moving the cursor keys while holding the shift key.
- Ctrl + C: Copy the marked text to the clipboard
- Shift + Del(ete): Delete the marked text and save it in the clipboard
- Ctrl + V: Insert text from the clipboard at the cursor position
- Del(ete): Delete the marked text

4.4 Program Section Code

A new DIN program is already provided with section codes. You can add new codes or delete existing ones, depending on your program requirements. A DIN program must contain at least the MACHINING and END section codes.

You can select further program section codes under the menu item "PAb" (**P**rogram section code) in the main menu, in the "Instr." menu or in the DIN PLUS Words selection box. The CNC PILOT inserts the section code at the correct position.



For more than one independent contour definition for drilling/milling, use the program section codes (FRONT END, REAR END, etc.) each time.

Overview of program section codes		
Program head		
	PROGRAMMKOPF [PROGRAM HEAD]	Page 136
	REVOLVER [TURRET]	Page 137
	SPANNMITTEL [CHUCKING EQUIPMENT]	Page 142
Contour definition		
	KONTUR [CONTOUR]	Page 143
	ROHTEIL [WORKPIECE BLANK]	Page 143
	FERTIGTEIL [FINISHED PART]	Page 143
	HILFSKONTUR [AUXILIARY CONTOUR]	Page 144
C-axis contours		
	STIRN [FRONT]	Page 144
	RUECKSEITE [REAR SIDE]	Page 144
	MANTEL [SURFACE]	Page 144
Workpiece machining		
	BEARBEITUNG [MACHINING]	Page 144
	ZUORDNUNG [ASSIGNMENT]	Page 144
	ENDE [END]	Page 144
Subroutines		
	UNTERPROGRAMM [SUBPROGRAM]	Page 145
	RETURN	Page 145
Others		
	CONST	Page 145

Example: Program section codes

```
. . .  [Sections of the contour
description]
ROHTEIL [WORKPIECE BLANK]
N1 G20 X100 Z220 K1

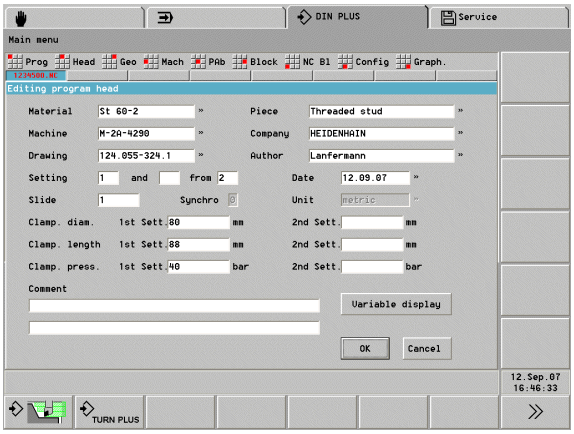
FERTIGTEIL [FINISHED PART]
N2 G0 X60 Z0
N3 G1 Z-70
. . .
STIRN Z-25 [FRONT]
N31 G308 P-10
N32 G402 Q5 K110 A0 Wi72 V2 XK0 YK0
N33 G300 B5 P10 W118 A0
N34 G309

STIRN Z0 [FRONT]
N35 G308 P-6
N36 G307 XK0 YK0 Q6 A0 K34.641
N37 G309
. . .
```

PROGRAM HEAD section

Instructions and information of the PROGRAM HEAD:

- **Slide**: The NC program is executed only for indicated slides
  - Enter 1 for $1
  - Enter 12 for $1 and $2
  - No entry: NC program is run on **every** slide.
- **Unit:**
  - Select dimensional system in millimeters or inches
  - No entry: The unit set in control parameter 1 is used.
- The other fields contain **organizational information** and **set-up information,** which do not influence the machining process.



Information contained in the program head is preceded by "#" in the DIN program.

In the "organization" (Transfer mode of operation) the entries of the "Drawing" field are shown in the list of NC main programs.



The unit can be programmed only when a new program is being created (set under PROGRAM HEAD). It is not possible to post-edit this entry.

Variable display:

Calling the display:

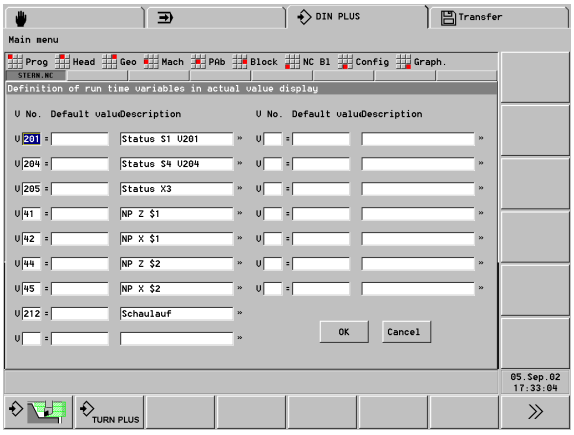
- Press the **Variable display** button in the "Editing program head" dialog box

In the dialog box you define up to 16 V variables that control the program process. In automatic mode and in the simulation you define whether the variables are to be asked for during program run. As an alternative, the program version is run with the default values.

For each variable you define:

- Variable number
- Default value (initialization value)
- Description (Text with which this variable is interrogated during program run)

The definition of the variable display is an alternative to programming with INPUTA/PRINTA commands.



TURRET section

The TURRET x program section (x: 1 to 6) defines the assignment of the tool carrier x. For every assigned turret pocket:

- The tool ID number is entered if the tool is in the database.
- The tool description is entered directly if it is a temporary tool. Temporary tools are not transferred into the database.

The following functions are available for editing the turret assignment:

- "Turret assignment" menu item: For each entry of this section you call the Tool dialog box and load a tool from the database, or describe the tool with the "Extended input" or as a "simple tool."
- "Set up tool list" menu item: As in the set-up function, the CNC PILOT provides the turret assignment of this NC program as a tool list for editing. In this function you use only tools from the database.

Tool description in the NC program:
Tools are usually described in the database and the tool ID number is entered as a reference in the NC program. As an alternative you can describe the tool in the NC program.

- Extended tool description:
  - The tool parameters correspond to the first dialog box of the tool editor.
  - There are no limits on the use of the tool.
  - Only the tool cutting edge is shown in the simulation.
  - If the ID number is entered, the data are taken into the database.
  - If the ID number is **not** entered, the data are not taken into the database.

- Simple tool description:
  - The tools are suitable only for simple traverse paths and turning cycles (G0...G3, G12, G13; G81...G88).
  - Contour follow-up is **not** taken into account.
  - The tool radius compensation is conducted.
  - Simple tools are **not** transferred into the database.



- If you **do not program the TURRET,** the tools entered in the tool list will be used.
- The names _SIM... and _AUTO... are reserved for temporary tools (simple tools and tools without ID number). Tool entries are valid as long as the NC program is activated in the simulation or in Automatic mode.

Example of turret table

`REVOLVER 1 [TURRET]`	
`T1 ID"342-300.1"`	Tool from the database
`T2 WT1 X50 Z50 R0.2 B6`	Simple tool description:
`T3 WT122 X15 Z150 H0 V4 R0.4 A93 C55 I9 K70`	Extended tool description without entry in the database
`T4 ID"ERW.1" WT112 X20 Z150 H2 V4 R0.8 A95 C80 B9 K70`	Extended tool description with entry in the database
`. . .`	

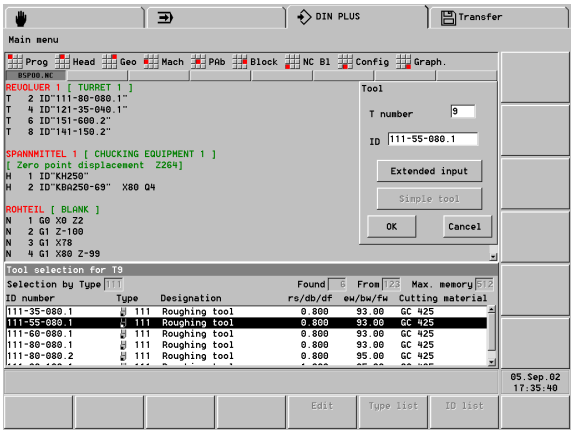
Editing the turret assignment

For every occupied turret pocket in the TURRET section:

- The tool ID number is entered if the tool is in the database.
- The tool description is entered directly if it applies to temporary tools.

Parameters of the Tool dialog box:

T number	Position on tool carrier
ID	Identification number (reference to database).
“Extended input” button	Switching to extended tool description
“Simple tool” button	Switching to simple tool description



Entering or editing tools:

- Select "Head > Turret assignment." The editor moves the cursor to the TURRET section.

Entering a tool:

- Position the cursor.

Ins

- Press the INS key. The editor opens the Tool dialog box.

- Edit the Tool dialog box

Editing the tool data:

- Position the cursor on the entry to be edited.
- Press RETURN or double-click with the left mouse key
- Edit the Tool dialog box.

Turret assignment from the tool database

From the Tool dialog box you have direct access to the database. You load the tool ID number.

Type list

- Press the soft key. The entries are listed sorted by tool type.

ID list

- Press the soft key. The entries are listed sorted by tool ID number.

- Place the cursor on the tool to be loaded.
- Press RETURN to load the ID number in the Tool dialog box

Edit the tool data

Edit

- Press the soft key. The CNC PILOT provides the data of the tool shown in the Tool dialog box for editing.

Loading a tool list

As of software version 625 952-04:

A tool list that has been set up in Machine mode can be loaded into the NC program.

- Place the cursor in the program section (TURRET 1, TURRET 2, PLATE MAGZN., etc.).
- Select “Head > Load list” in the main menu.

The CNC PILOT transfers the selected turret or magazine list into the NC program. If you have already entered tools, they will be deleted after a confirmation prompt.

Editing the turret assignment directly

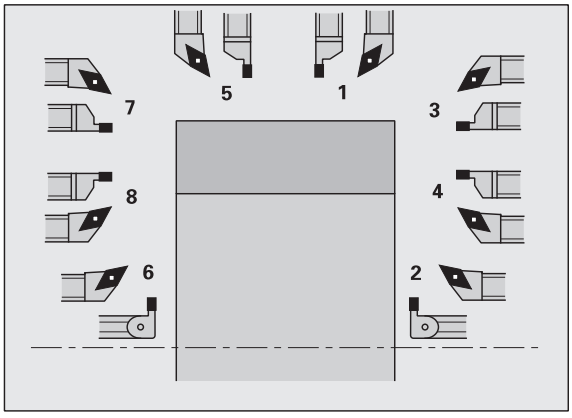
Extended tool description:

- Press the “Extended input” button (Tool dialog box). The editor opens the “Tool type” dialog box.
- Enter the tool type. The editor opens the dialog box of the selected tool type.
- Enter the tool data (the data correspond to the first dialog box of the tool database).

Simple tool description:

- Press the “Simple tool” button (Tool dialog box). The editor opens the “Tool type” dialog box.
- Enter the tool type. The editor opens the Tool dialog box.
- Enter the tool data.

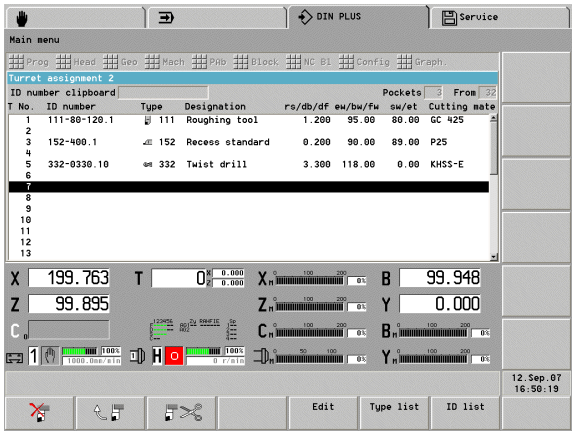
Simple tools		
Dialog box	**NC pgm**	**Meaning**
Tool type	WT	Tool type and machining direction (see illustration):
X dimension (xe)	X	Setup dimension
Y dimension (ye)	Y	Setup dimension
Z dimension (ze)	Z	Setup dimension
Radius R (rs)	R	Cutting radius of turning tools
Tool width B (sb)	B	Cutting width of recessing and button tools
Diameter I (df)	I	Milling or drilling diameter



Turret assignment as tool list

In the “Set up tool list” function, the CNC PILOT provides the turret assignment as a tool list for editing. The procedure is the same as in the “set up list” function (see “Setting up a tool list” on page 68)

- Select “Head > Set up tool list”
- Place the cursor on the position to be edited
- Edit the tool entry



Soft keys	
	Delete tool
	Take the tool from the clipboard
	Delete the tool and put it in the clipboard
Editing	Edit the tool parameters
Type list	Entries in the tool database—sorted by type
ID list	Entries in the tool database—sorted by ID number

CHUCKING EQUIPMENT section

The CHUCKING EQUIPMENT x program section (x: 1 to 4) defines spindle assignment x. Using the identification numbers of chuck, jaws and adapters (lathe center, etc.), you create the chucking equipment table.

Parameters of the chucking equipment dialog box

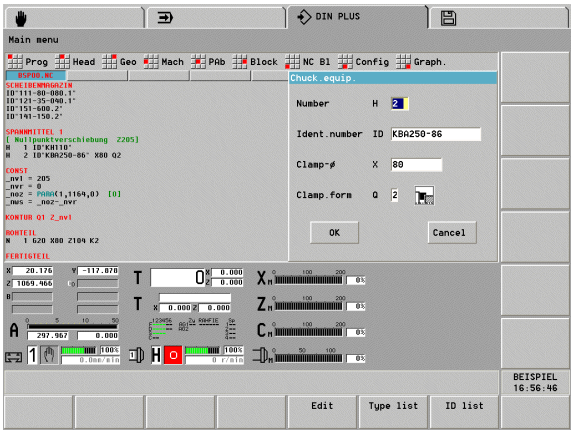
H Clamp number (reference for G65)

- H=1: Chuck
- H=2: Chuck jaw
- H=3: Chuck supplement—spindle-side
- H=4: Chuck supplement—tailstock side

ID Identification number of chucking equipment (reference to database).

X clamping diameter of chuck jaw

Q Clamping form of jaws (see G65)



Example: Chucking equipment table

```
SPANNMITTEL 1 [CHUCKING EQUIPMENT]
H1 ID"KH250"
[Chuck]
H2 ID"KBA250-77"
[Chuck jaws]
. . .
```

The chucking equipment table is evaluated in the simulation (G65). It has no influence on the machining process.

Entering the chucking equipment data:

- Select “Head > Chucking equipment.” The CNC PILOT positions the cursor to the CHUCKING EQUIPMENT section.
- Position the cursor.
- Press the INS key: The editor opens the “Slide number” dialog box.
- Edit the dialog box

Editing the chucking equipment data:

- Position the cursor on the chucking equipment
- Press ENTER
- Edit the chucking equipment dialog box

CONTOUR section

The CONTOUR program section assigns the following workpiece blank and finished part description to the contour "number x." The control manages up to four contours (workpieces) in one NC program.

A G99 in the machining section assigns the contour to a slide or spindle.

Parameters

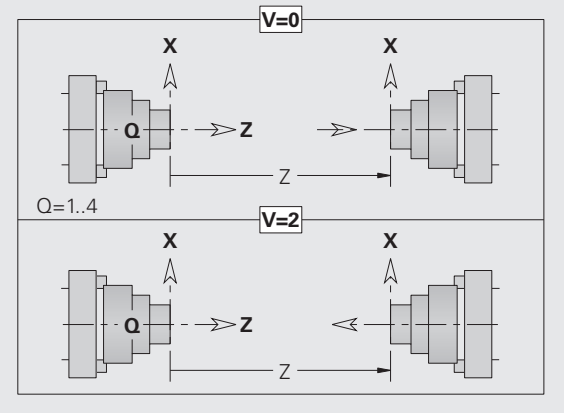
Q Number of the contour (1 to 4)

X Zero point shift (diameter value)

Z Datum shift

V Position of the coordinate system

- V=0: The coordinate system applies
- V=2: Mirrored machine coordinate system (Z direction always away from the spindle)



If a workpiece is machined in the NC program the CONTOUR section and G99 are not required.

Example: Contour and G99

```
PROGRAMMKOPF [PROGRAM HEAD]
...
CONTOUR Q1 X0 Z600                [contour 1]
ROHTEIL [WORKPIECE BLANK]
...
FERTIGTEIL [FINISHED PART]
. . .
CONTOUR Q2 X0 Z900 V2             [contour 2]
ROHTEIL [WORKPIECE BLANK]
. . .
FERTIGTEIL [FINISHED PART]
. . .
BEARBEITUNG [MACHINING]
. . .
N.. G99 Q2 D4
. . .
```

BLANK section

In the BLANK section you describe the contour of the workpiece blank.

FINISHED PART section

In the FINISHED PART section you describe the contour of the finished part. Within the FINISHED PART section you use additional codes such as FRONT, SURFACE, etc.

AUXILIARY CONTOUR section

In the AUXILIARY CONTOUR section you describe the auxiliary contours of the turning contour.

FRONT section

In the FRONT section you describe the contours of the front face.

Parameters

Z Position of front face contour

REAR SIDE section

In the REAR SIDE section you describe the contours of the back face.

Parameters

Z Position of the rear-face contour

SURFACE section

In the SURFACE section you describe the cylindrical-surface contours.

Parameters

X Reference diameter of lateral-surface contours.

MACHINING section

In the MACHINING section you program the machining operations. This code **must** be included.

END code

The END section concludes the NC program. This code **must** be programmed. It replaces M30.

ZUORDNUNG [ASSIGNMENT] instruction $..

The ASSIGNMENT instruction assigns the subsequent machining operations to the given slides. If more than one slide is listed the NC blocks are executed on the given slides.

If in addition a slide code is shown, the slides shown under $.. are used.

Parameters

Slide	Slide number(s)

SUBPROGRAM section

If you define a subprogram within your NC program (within the same file), it is designated with SUBPROGRAM, followed by the name of the subprogram (max. 8 characters).

RETURN code

The RETURN code concludes the subprogram.

CONST code

In the CONST section of the program you define constants. You use constants for the definition of:

- a value
- a # variable
- a V variable

You enter the value directly or you calculate it. If you use constants in the calculation you must first define them.

The length of the constant name must not exceed 16 characters.

Example: CONST

```
CONST
[_nvr: Zero point shift]
[_noz: Zero point offset]
[_nws: Shift]
_nvr = 0
_noz = PARA(1,1164,0)
_nws = _noz-_nvr
_lg_roht = 1                    [Variable #1]
_posbeginn = 178              [Variable V178]
. . .
CONTOUR Q4 X0 Z_nws V2
ROHTEIL [WORKPIECE BLANK]
N 3 #_lg_roht=270
N 1 G20 X120 Z#_lg_roht K2
. . .
BEARBEITUNG [MACHINING]
. . .
N 6 G0 X{V_posbeginn}
. . .
```

4.5 Definition of Workpiece Blank

Chuck piece: bar/tube G20-Geo

G20 defines the contour of a cylinder/hollow cylinder.

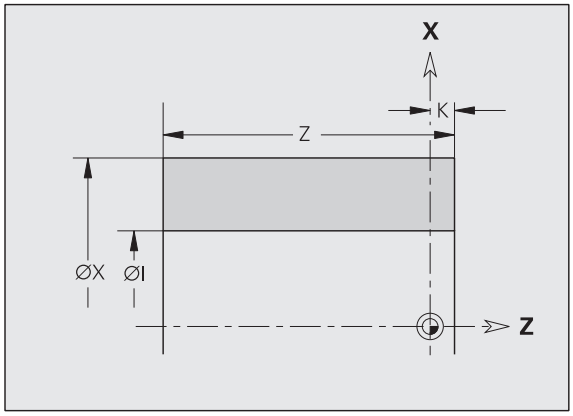
Parameters

X
- Cylinder/hollow cylinder diameter
- Diameter of circumference of a polygonal blank

Z Length of the blank

K Right edge (distance between workpiece zero point and right edge)

I Inside diameter of hollow cylinders



Example: G20-Geo

```
. . .
ROHTEIL [WORKPIECE BLANK]
N1 G20 X80 Z100 K2 I30    [hollow cylinder]
. . .
```

Casting G21-Geo

G21 generates the contour of the blank part from the contour of the finished part—plus the equidistant oversize P.

Parameters

P Equidistant oversize (reference: finished part contour)

Q Hole Yes/No (default: 0)
- Q=0: Without hole
- Q=1: With hole

Example: G21-Geo

```
. . .
ROHTEIL [WORKPIECE BLANK]
N1 G21 P5 Q1                    [cast blank]
. . .
FERTIGTEIL [FINISHED PART]
N2 G0 X30 Z0
N3 G1 X50 B-2
N4 G1 Z-40
N5 G1 X65
N6 G1 Z-70
. . .
```

4.6 Basic Contour Elements

Starting point of turning contour G0-Geo

G0 defines the starting point of a turning contour.

Parameters

X Contour starting point (diameter value)

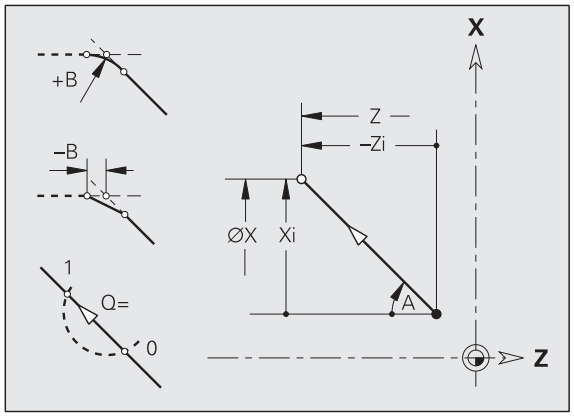
Z Contour starting point

Example: G0-Geo

```
. . .
FERTIGTEIL [FINISHED PART]
N2 G0 X30 Z0     [starting point of contour]
N3 G1 X50 B-2
N4 G1 Z-40
N5 G1 X65
N6 G1 Z-70
. . .
```

Line segment in a contour G1-Geo

G1 defines a line segment in a turning contour.



Parameters

X End point of contour element (diameter value)

Z End point of contour element

A Angle to rotary axis (for angle direction see graphic support window)

Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E Special feed rate for the chamfer/rounding arc during the finishing cycle (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)

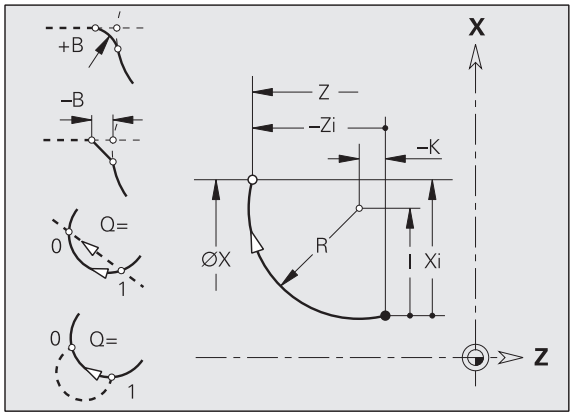
Example: G1-Geo

`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`N2 G0 X0 Z0`	Starting point
`N3 G1 X50 B-2`	Perpendicular line with chamfer
`N4 G1 Z-20 B2`	Horizontal line with radius
`N5 G1 X70 Z-30`	Oblique cut with absolute target coordinates
`N6 G1 ZI-5`	Horizontal line segment, incremental
`N7 G1 XI10 A30`	Incremental and angle
`N8 G1 X92 ZI-5`	Incremental and absolute mixed
`N9 G1 X? Z-80`	Calculate the X coordinate
`N10 G1 X100 Z-100 A10`	End point and angle with unknown starting point
`. . .`	

Circular arc of turning contour G2/G3-Geo

G2/G3 defines a circular arc in a contour with **incremental** center dimensioning. Direction of rotation (see help graphic):

- G2: In clockwise direction
- G3: In counterclockwise direction



Parameters

X End point of contour element (diameter value)

Z End point of contour element

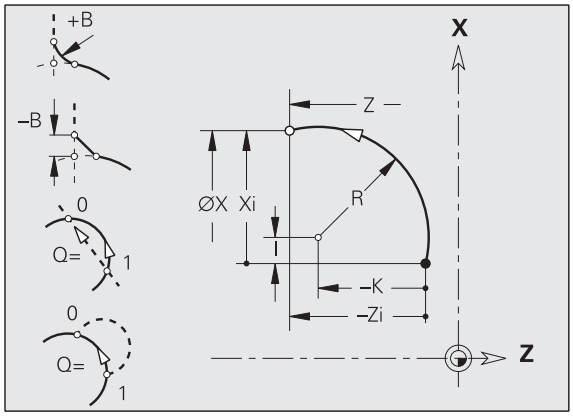
I Center (distance from starting point to center as radius)

K Center (distance from starting point to center)

R Radius

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection



Parameters

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E Special feed rate for the chamfer/rounding arc during the finishing cycle (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)



Programming X, Z: Absolute, incremental, modal or "?"

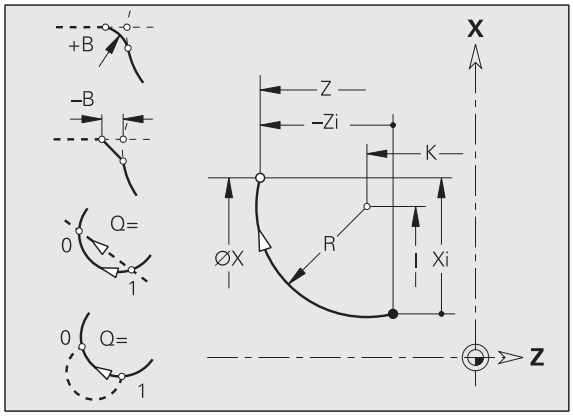
Example: G2-, G3-Geo

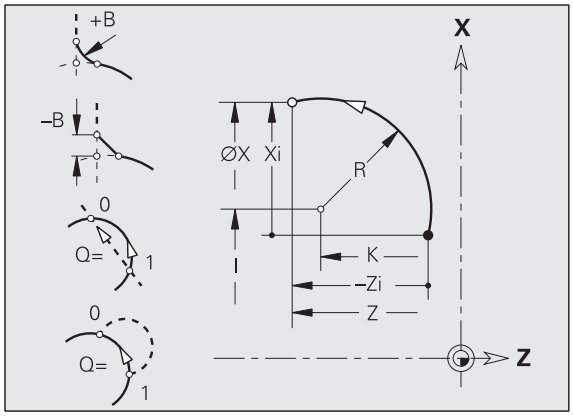
`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`N1 G0 X0 Z-10`	
`N2 G3 X30 Z-30 R30`	Target point and radius
`N3 G2 X50 Z-50 I19.8325 K-2.584`	Target point and center, incremental
`N4 G3 XI10 ZI-10 R10`	Target point (incremental) and radius
`N5 G2 X100 Z? R20`	Unknown target point coordinate
`N6 G1 XI-2.5 ZI-15`	
`. . .`	

Circular arc of turning contour G12/G13-Geo

G12/G13 defines a circular arc in a contour with **absolute** center dimensioning. Direction of rotation (see help graphic):

- G12: In clockwise direction
- G13: In counterclockwise direction





Parameters

X End point of contour element (diameter value)

Z End point of contour element

I Center (radius dimension)

K Center

R Radius

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E Special feed rate for the chamfer/rounding arc during the finishing cycle (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)

Programming X, Z: Absolute, incremental, modal or "?"

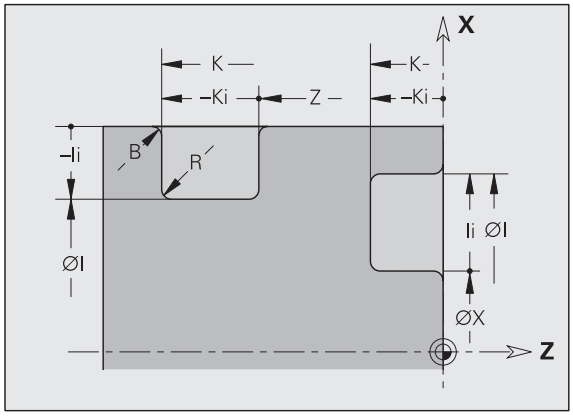
Example: G12-, G13-Geo

. . .	
FERTIGTEIL [FINISHED PART]	
N1 G0 X0 Z-10	
. . .	
N7 G13 XI-15 ZI15 R20	Target point (incremental) and radius
N8 G12 X? Z? R15	Only the radius is known
N9 G13 X25 Z-30 R30 B10 Q1	Rounding arc in transition and selection of intersections
N10 G13 X5 Z-10 I22.3325 K-12.584	Target point and center, absolute
. . .	

4.7 Contour Form Elements

Recess (standard) G22-Geo

G22 defines a recess on the previously programmed paraxial reference element.



Parameters

X Starting point of recess on the face (diameter)

Z Starting point of recess on the lateral surface

I Inside corner (diameter value)

- Recessing on face: End point of the recess
- Recess on lateral surface: Recess floor

K Inside corner

- Recess on face: Recess base
- Recessing on lateral surface: End point of the recess

Ii Inside corner—incremental (pay attention to algebraic sign!)

- Recess on face: Recess width
- Recess on lateral surface: Recess depth

Ki Inside corner—incremental (pay attention to algebraic sign!)

- Recess on face: Recess depth
- Recess on lateral surface: Recess width

B Outside radius/chamfer at both sides of the recess (default: 0)

- B>0: Rounding radius
- B<0: Chamfer width

R Inside radius in both corners of recess (default: 0)

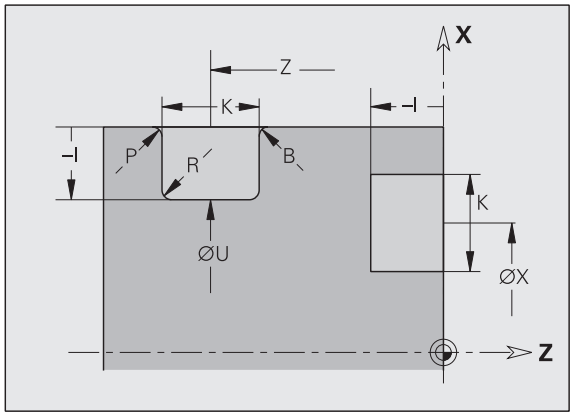
Program either X or Z.

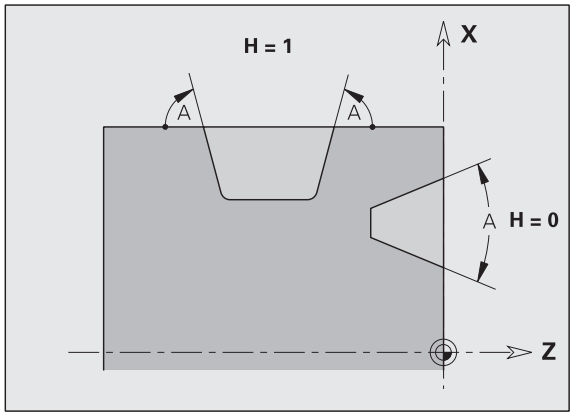
Example: G22-Geo

`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`N1 G0 X40 Z0`	
`N2 G1 X80`	
`N3 G22 X60 I70 KI-5 B-1 R0.2`	Recess on face, depth is incremental
`N4 G1 Z-80`	
`N5 G22 Z-20 I70 K-28 B1 R0.2`	Longitudinal recess, width is absolute
`N6 G22 Z-50 II-8 KI-12 B0.5 R0.3`	Longitudinal recess, width is incremental
`N7 G1 X40`	
`N8 G1 Z0`	
`N9 G22 Z-38 II6 K-30 B0.5 R0.2`	Longitudinal recess, inside
`. . .`	

Recess (general) G23-Geo

G23 defines a recess on the previously programmed linear reference element. On the lateral surface the reference element can be diagonal.





Parameters

H Type of recess (default: 0)

- H=0: Symmetrical recess
- H=1: Relief turn

X Center point of recess on the face (diameter)

Z Center point of recess on the lateral surface

I Recess depth and recess position

- I>0: Recess at the right of the reference element
- I<0: Recess at the left of the reference element

K Recess width (without chamfer/rounding arc)

U Recess diameter (diameter of recess base). Use U only if the reference element runs parallel to the Z axis

A Recess angle (default: 0)

- H=0: 0° <= A < 180° (angle between recess edges)
- H=1: 0° < A <= 90° (angle between reference line and recess edge)

Parameters

B Outside radius/chamfer at corner near the starting point (default: 0)

- B>0: Rounding radius
- B<0: Chamfer width

P Outside radius/chamfer at corner far from the starting point (default: 0)

- P>0: Radius of the rounding arc
- P<0: Chamfer width

R Inside radius in both corners of recess (default: 0)



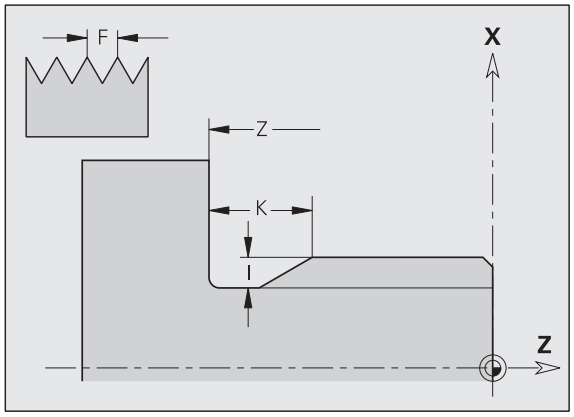
The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.

Example: G23-Geo

`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`N1 G0 X40 Z0`	
`N2 G1 X80`	
`N3 G23 H0 X60 I-5 K10 A20 B-1 P1 R0.2`	Recess on face, depth is incremental
`N4 G1 Z-40`	
`N5 G23 H1 Z-15 K12 U70 A60 B1 P-1 R0.2`	Longitudinal recess, width is absolute
`N6 G1 Z-80 A45`	
`N7 G23 H1 X120 Z-60 I-5 K16 A45 B1 P-2 R0.4`	Longitudinal recess, width is incremental
`N8 G1 X40`	
`N9 G1 Z0`	
`N10 G23 H0 Z-38 I-6 K12 A37.5 B-0.5 R0.2`	Longitudinal recess, inside
`. . .`	

Thread with undercut G24-Geo

G24 defines a linear basic element with a longitudinal thread and subsequent thread undercut (DIN 76). The thread is an outside or inside thread (metric ISO fine-pitch thread DIN 13 Part 2, Series 1).



Parameters

F Thread pitch

I Undercut depth (radius)

K Width of undercut

Z End point of the undercut

- Program G24 only if the thread is cut in the definition direction of the contour.
- The thread is machined with G31.

Example: G24-Geo

`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`N1 G0 X40 Z0`	
`N2 G1 X40 B-1.5`	Starting point for thread
`N3 G24 F2 I1.5 K6 Z-30`	Thread with undercut
`N4 G1 X50`	Next transverse element
`N5 G1 Z-40`	
`. . .`	

Undercut contour G25-Geo

G25 generates the following undercut contours in paraxial inside contour corners. Program G25 after the first axis-parallel element. You specify the undercut type in parameter H.

Undercut type U (H=4)

Parameters

H Undercut type U: H=4

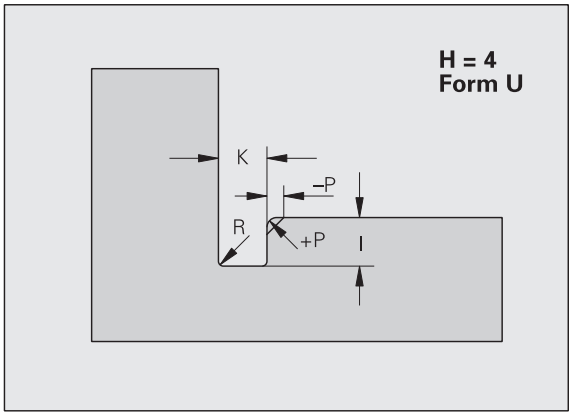
I Undercut depth (radius)

K Width of undercut

R Inside radius in both corners of recess (default: 0)

P Outside radius/chamfer (default: 0)

- P>0: Radius of the rounding arc
- P<0: Chamfer width



Example: Call G25-Geo type U

```
. . .
N.. G1 Z-15              [longitudinal element]
N.. G25 H4 I2 K4 R0.4 P-0.5          [Type U]
N.. G1 X20               [transverse element]
. . .
```

Undercut DIN 509 E (H=0.5)

Parameters

H Undercut type DIN 509 E: H=0 or H=5

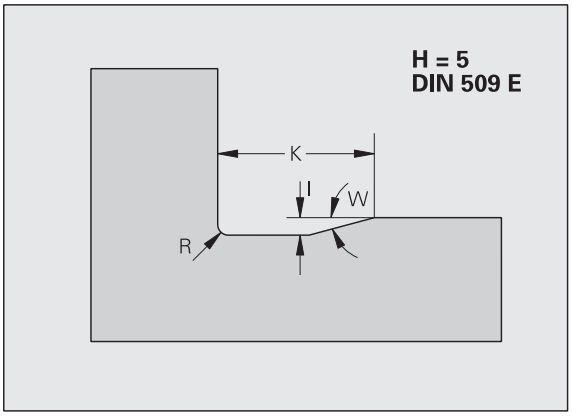
I Undercut depth (radius)

K Width of undercut

R Undercut radius (in both corners of the undercut)

W Undercut angle

The CNC PILOT uses the diameter to calculate the parameters that you do not define.



Example: Call G25-Geo DIN 509 E

```
. . .
N.. G1 Z-15          [longitudinal element]
N.. G25 H5                      [DIN 509 E]
N.. G1 X20           [transverse element]
. . .
```

Undercut DIN 509 F (H=6)

Parameters

H Undercut type DIN 509 F: H=6

I Undercut depth (radius)

K Width of undercut

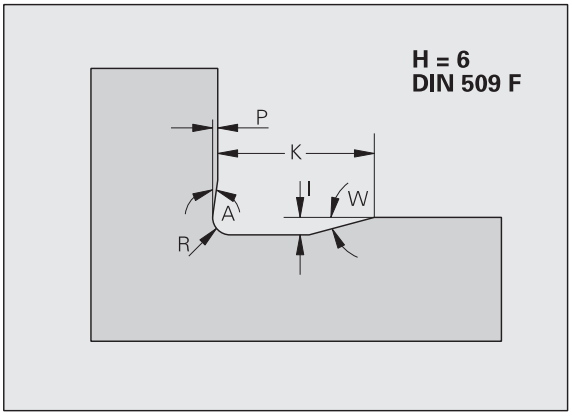
R Undercut radius (in both corners of the undercut)

P Face depth

W Undercut angle

A Transverse angle

The CNC PILOT uses the diameter to calculate the parameters that you do not define.



Example: Call G25-Geo DIN 509 F

```
. . .
N.. G1 Z-15          [longitudinal element]
N.. G25 H6                      [DIN 509 F]
N.. G1 X20           [transverse element]
. . .
```

Undercut DIN 76 (H=7)

Parameters

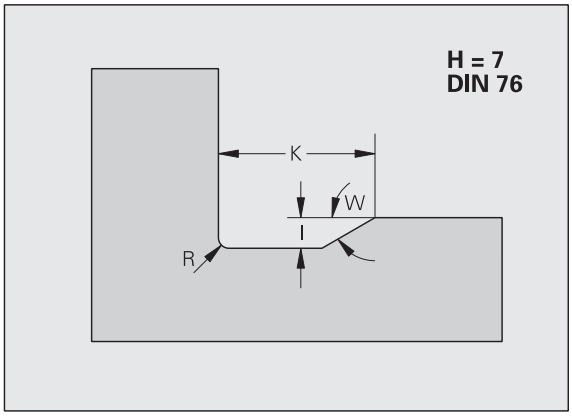
H Undercut type DIN 76: H=7

I Undercut depth (radius)

K Width of undercut

R Undercut radius in both corners of the undercut (default: R=0.6*I)

W Undercut angle (default: 30°)



Example: Call G25-Geo DIN 76

```
. . .
N.. G1 Z-15              [longitudinal element]
N.. G25 H7 I1.5 K7                      [DIN 76]
N.. G1 X20               [transverse element]
. . .
```

Undercut type H (H=8)

If you do not enter W, the angle will be calculated on the basis of K and R. The end point of the undercut is then located at the "contour corner."

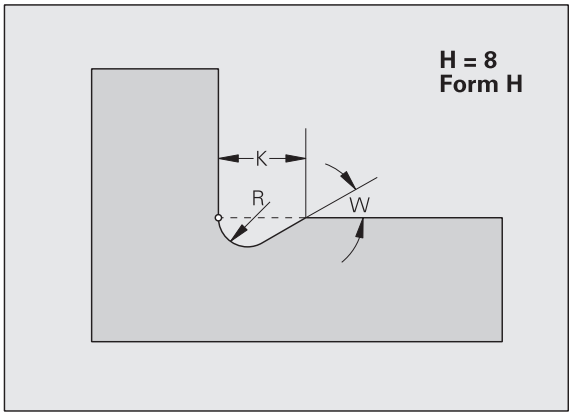
Parameters

H Undercut type H: H=8

K Width of undercut

R Undercut radius—no value: The circular element is not machined

W Plunge angle—no value: W is calculated



Example: Call G25-Geo type H

```
. . .
N.. G1 Z-15              [longitudinal element]
N.. G25 H8 K4 R1 W30                    [type H]
N.. G1 X20               [transverse element]
. . .
```

Undercut type K (H=9)

Parameters

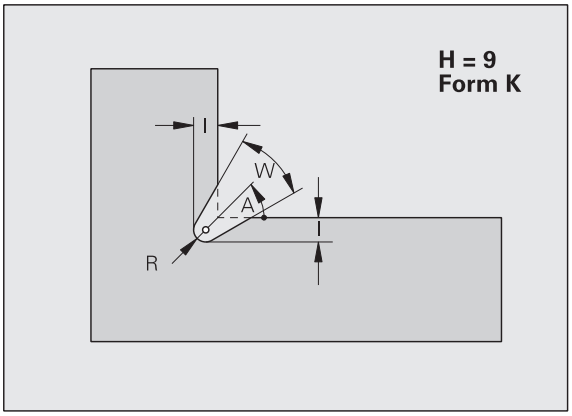
H Undercut type K: H=9

I Undercut depth

R Undercut radius—no value: The circular element is not machined

W Undercut angle

A Angle to longitudinal axis (default: 45°)



Example: Call G25-Geo type K

```
. . .
N.. G1 Z-15              [longitudinal element]
N.. G25 H9 I1 R0.8 W40                 [type K]
N.. G1 X20               [transverse element]
. . .
```

Thread (standard) G34-Geo

G34 defines a simple or an interlinked external or internal thread (metric ISO fine-pitch thread DIN 13 Series 1). The CNC PILOT calculates all the required values.

Parameters

F Thread pitch (default: pitch from the standard table

Threads are interlinked by programming several G01/G34 blocks after each other.

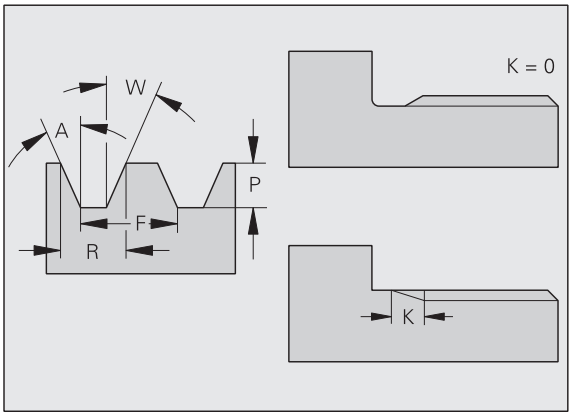
- You need to program a linear contour element as a reference before G34 or in the NC block containing G34.
- Machine the thread with G31.

Example: G34

```
. . .
FERTIGTEIL [FINISHED PART]
N1 G0 X0 Z0
N2 G1 X20 B-2
N3 G1 Z-30
N4 G34                                 [metric ISO]
N5 G25 H7 I1.7 K7
N6 G1 X30 B-1.5
N7 G1 Z-40
N8 G34 F1.5  [metric ISO fine-pitch thread]
N9 G25 H7 I1.5 K4
N10 G1 X40
N11 G1 Z-60
. . .
```

Thread (general) G37-Geo

G37 defines the different types of thread. Multi-start threads and concatenated threads are possible. Threads are concatenated by programming several G01/G34 blocks after each other.



Parameters

Q Type of thread (default: 1)

- Q=1: Metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
- Q=2: Metric ISO thread (DIN 13 Part 1, Series 1)
- Q=3: Metric ISO tapered thread (DIN 158)
- Q=4: Metric ISO tapered fine-pitch thread (DIN 158)
- Q=5: Metric ISO trapezoid thread (DIN 103 Part 2, Series 1)
- Q=6: Flat metric trapezoid thread (DIN 380 Part 2, Series 1)
- Q=7: Metric buttress thread (DIN 513 Part 2, Series 1)
- Q=8: Cylindrical round thread (DIN 405 Part 1, Series 1)
- Q=9: Cylindrical Whitworth thread (DIN 11)
- Q=10: Tapered Whitworth thread (DIN 2999)
- Q=11: Whitworth pipe thread (DIN 259)
- Q=12: Nonstandard thread
- Q=13: UNC US coarse thread
- Q=14: UNF US fine-pitch thread
- Q=15: UNEF US extra-fine-pitch thread
- Q=16: NPT US taper pipe thread
- Q=17: NPTF US taper dryseal pipe thread
- Q=18: NPSC US cylindrical pipe thread with lubricant
- Q=19: NPFS US cylindrical pipe thread without lubricant

F Thread pitch

- Required for Q=1, 3 to 7, 12.
- For other thread types, F is calculated from the diameter if it was not programmed.

P Thread depth—enter only for Q=12

K Run-out length for threads without undercut (default: 0)

D Reference point (default: 0)

- D=0: Runout of thread at the end of the reference element
- D=1: Runout of thread at the beginning of the reference element

H Number of thread turns (default: 1)

A Edge angle at left—enter only for Q=12

W Edge angle at right—enter only for Q=12

R Thread width—enter only for Q=12

E Variable pitch (default: 0)

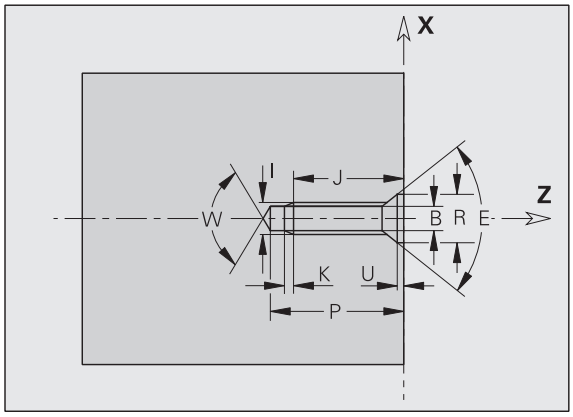
Increase/decrease the pitch per revolution by E.

- Before G37, program a linear contour element as a reference.
- Machine the thread with G31.
- For standard threads, the parameters P, R, A and W are defined by the CNC PILOT.
- Use Q=12 if you wish to use individual parameters.

Danger of collision!
The thread is generated to the length of the reference element. Another linear element without undercut is to be programmed as overrun.

Hole (centric) G49-Geo

G49 defines a single hole with countersink and thread **at the turning center** (front or rear face). The G49 hole is a form element, not part of the contour.



Parameters

Z Starting position for hole (reference point)

B Hole diameter

P Depth of hole (excluding point)

W Point angle (default: 180°)

R Sinking diameter

U Sinking depth

E Sinking angle

I Thread diameter

J Thread depth

K Thread runout length

F Thread pitch

V Left-hand or right-hand thread (default: 0)

- V=0: Right-hand thread
- V=1: Left-hand thread

A Angle corresponding to the position of the hole (default: 0)

- A=0°: Front face
- A=180°: Rear side

O Centering diameter

- Program G49 in the FINISHED PART segment, not in FRONT or REAR SIDE.
- Machine the G49 hole with G71...G74.

4.8 Attributes for Contour Description

Overview of attributes for contour description		
G7	Precision stop ON	Page 164
G8	Precision stop OFF	Page 164
G9	Precision stop blockwise	Page 164
G10	Influences the finishing feed rate for basic contour elements of the entire contour.	Page 164
G38	Influences finishing feed rate for basic contour elements block by block	Page 165
G39	Applies only for **form elements**: ■ Influences finishing feed rate ■ Additive compensation ■ Equidistant oversizes	Page 165
G52	Blockwise equidistant oversize	Page 166
G95	Defines finishing feed rate for total contour	Page 166
G149	Additive compensation values for basic contour elements	Page 167



- G10, G38, G52, G95 and G149 Geo apply for basic contour elements (G1, G2, G3, G12 and G13 Geo), **not** for chamfers/rounding arcs that are programmed in connection with basic contour elements.
- The attributes for contour description influence the finishing feed rate of the Cycles G869 and G890, not the finishing feed rate in recessing cycles.

Precision stop

Precision stop ON G7-Geo

G7 switches precision stop on. It is a modal function. The block with G7 is run **with** precision stop. The CNC PILOT does not run the following block until the tool reaches the position tolerance window around the end point (for more on the tolerance window, see MP 1106, 1156, ...).



Precision stop is used for basic contour elements that are executed with G890 or G840.

Precision stop OFF G8-Geo

G8 switches precision stop off. The block with G8 is run **without** precision stop.

Blockwise precision stop G9 Geo

G9 activates a precision stop for the block in which G9 is programmed.

Surface roughness G10-Geo

G10 influences the finishing feed rate of G890. The surface roughness (peak-to-valley height) applies only to basic contour elements.

Parameters

H Type of surface roughness (see also DIN 4768)

- H=1: General surface roughness (profile depth) Rt1
- H=2: Surface roughness Ra
- H=3: Surface roughness Rz

RH Surface roughness (µm, inch mode: µinch)



- G10 Geo is modal.
- G95 Geo or G10 Geo without parameters switch the surface roughness off.
- G10 RH... (without “H”) overwrites the surface roughness blockwise.
- G38 Geo overwrites the surface roughness block by block.

Feed rate reduction factor G38-Geo

G38 activates the special feed rate for the finishing cycle G890. The "special feed rate" applies only to basic contour elements.

Parameters

E Special feed factor (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)



- G38 is a non-modal function.
- Program G38 **before** the contour element for which it is intended.
- G38 **replaces** another special feed rate or programmed surface roughness.

Attributes for superimposed elements G39-Geo

G39 influences the finishing feed rate of G890 with the form elements:

- Chamfers/rounding arcs (for connecting basic elements)
- Undercuts
- Recesses

Affected machining: Special feed rate, surface roughness, additive D compensation, equidistant oversizes.

Parameters

F Feed per revolution

V Type of surface roughness (see also DIN 4768)

- V=1: General surface roughness (profile depth) Rt1
- V=2: Surface roughness Ra
- V=3: Surface roughness Rz

RH Surface roughness (µm, inch mode: µinch)

D Number of the additive compensation (901 <= D <= 916)

P Oversize (radius)

H P applies as an absolute or additive value (default: 0)

- H=0: P replaces G57-/G58 oversizes
- H=1: P is added to G57-/G58

E Special feed factor (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)

- Use surface roughness (V, RH), finishing feed rate (F) and special feed rate ("E") alternately!
- G39 is a non-modal function.
- Program G39 **before** the contour element for which it is intended.
- G50 preceding a cycle (MACHINING section) cancels a finishing oversize programmed for that cycle with G39.

Blockwise oversize G52-Geo

G52 defines an equidistant oversize that is taken into consideration in G810, G820, G830, G860 and G890.

Parameters

P Oversize (radius)

H P applies as an absolute or additive value (default: 0)

- H=0: P replaces G57-/G58 oversizes
- H=1: P is added to G57-/G58

- G52 is a non-modal function.
- Program G52 **in** the NC block for which it is intended.
- G50 preceding a cycle (MACHINING section) cancels a finishing oversize programmed for that cycle with G52.

Feed per revolution G95-Geo

G95 influences the finishing feed rate of G890.

Parameters

F Feed per revolution

- Use surface roughness and finishing feed rate alternately.
- The G95 finishing feed rate replaces a finishing feed rate defined in the machining program.
- G95 is a modal function.
- G10 resets a finishing feed rate set with G95.

Additive compensation G149-Geo

G149 followed by a D number activates/deactivates an additive compensation function. The CNC PILOT manages the 16 tool-independent compensation values in the setup parameter10.

Parameters

D Additive compensation (default: D900)

- D=900: Deactivates the additive compensation
- D=901 to 916: Activates the additive compensation D



- Note the direction of contour description.
- Additive compensation is effective from the block in which G149 is programmed.
- Additive compensation remains in effect until:
  - the next G149 D900,
  - up to the end of the finished part description.

4.9 C-Axis Contours—Fundamentals

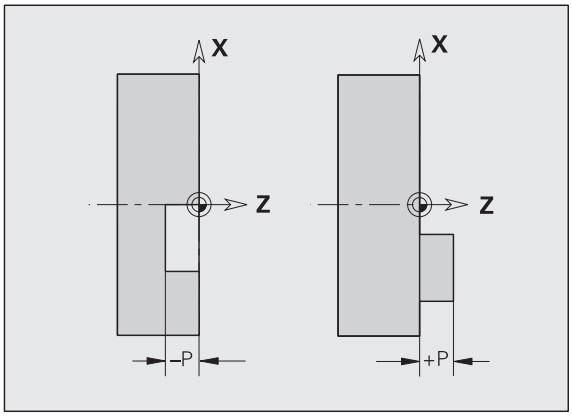
Milling contour position

Define the reference plane or the reference diameter in the section code. Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With **depth P** programmed in the previous G308 cycle.
- Alternatively on figures: Cycle parameter **depth P**.

The **algebraic sign of "P"** defines the position of the milling contour:

- P<0: Pocket
- P>0: Island



Milling contour position			
Section	**P**	**Surface**	**Milling floor**
STIRN [FRONT]	P<0	Z	Z+P
	P>0	Z+P	Z
RUECKSEITE [REAR SIDE]	P<0	Z	Z–P
	P>0	Z–P	Z
MANTEL [SURFACE]	P<0	X	X+(P*2)
	P>0	X+(P*2)	X

- X: Reference diameter from the section code
- Z: Reference plane from the section code
- P: Depth from G308 or from cycle parameter

The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.

Contours in more than one plane (hierarchically nested contours):

- A plane begins with G308 and ends with G309.
- G308 defines a new reference plane/reference diameter. The first G308 uses the reference plane defined in the section code. Each following G308 defines a new plane. Calculation:
  New reference plane = Reference plane + P (from previous G308)
- G309 switches back to the previous reference plane.

Start pocket/island G308-Geo

G308 defines a new reference plane / reference diameter in hierarchically nested contours.

Parameters

P Depth for pocket, height for islands

End pocket/island G309-Geo

G309 defines the end of a reference plane. Every reference plane defined with G308 **must** be ended with G309 (see "Milling contour position" on page 168).

Example for G308/G309

`. . .`	
`FERTIGTEIL [FINISHED PART]`	
`. . .`	
`STIRN Z0 [FRONT]`	Define reference plane
`N7 G308 P-5`	Beginning of rectangle with depth of –5
`N8 G305 XK-5 YK-10 K50 B30 R3 A0`	Rectangle
`N9 G308 P-10`	Beginning of "full circle in rectangle" with depth –10
`N10 G304 XK-3 YK-5 R8`	Full circle
`N11 G309`	End of full circle
`N12 G309`	End of rectangle
`MANTEL X100 [SURFACE]`	Define reference diameter
`N13 G311 Z-10 C45 A0 K18 B8 P-5`	Linear slot with depth –5
`. . .`	

Circular pattern with circular slots

For circular slots in circular patterns you program the pattern positions, the center of curvature, the curvature radius and the position of the slots.

The following rules apply in DIN PLUS and TURN PLUS for positioning and orienting the slots:

- Slots are arranged at the distance of the **pattern radius** about the **pattern center** if
  - Pattern center = center of curvature **and**
  - Pattern radius = curvature radius
- Slots are arranged at the distance of the **pattern radius + curvature radius** about the **pattern center** if
  - Pattern center <> center of curvature **or**
  - Pattern radius <> curvature radius

In addition, the position influences the arrangement of the slots:

- **Normal position:** The starting angle of the slot applies as a **relative value** to the pattern position. The starting angle is added to the pattern position.
- **Original position:** The starting angle of the slot applies as an **absolute value** to the pattern position.

The following examples show the programming of a circular pattern with circular slots:

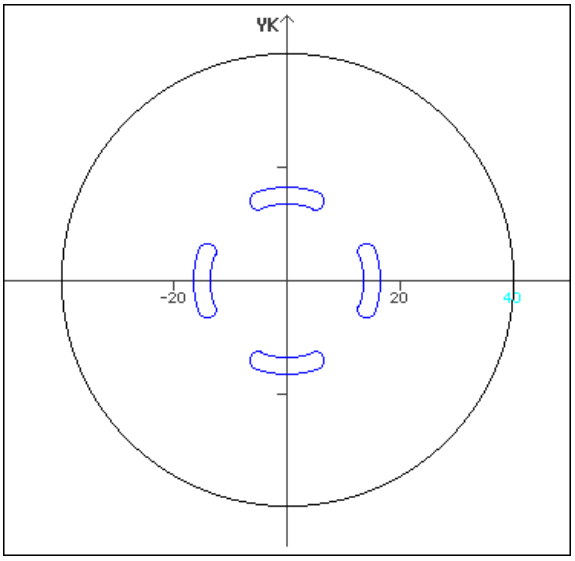
Slot centerline as reference and normal position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the pattern radius about the pattern center.

Example: Slot centerline as reference, normal position



`N.. G402 Q4 K30 A0 XK0 YK0 H0`	Circular pattern, normal position
`N.. G303 I0 J0 R15 A-20 W20 B3 P1`	Circular slot

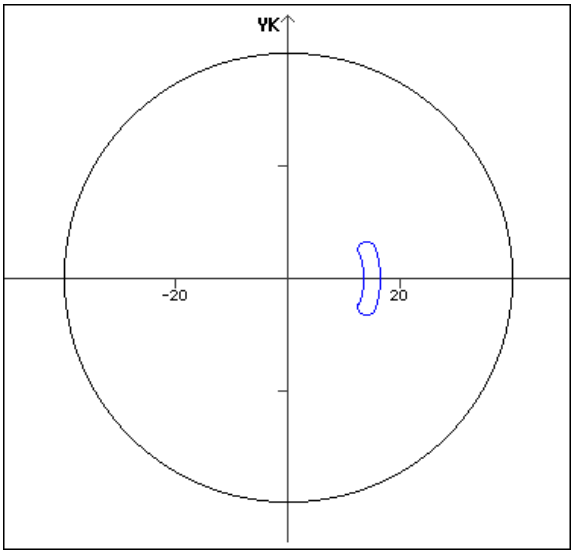
Slot centerline as reference and original position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange all slot at the same position.

Example: Slot centerline as reference, original position



`N.. G402 Q4 K30 A0 XK0 YK0 H1`	Circular pattern, original position
`N.. G303 I0 J0 R15 A-20 W20 B3 P1`	Circular slot

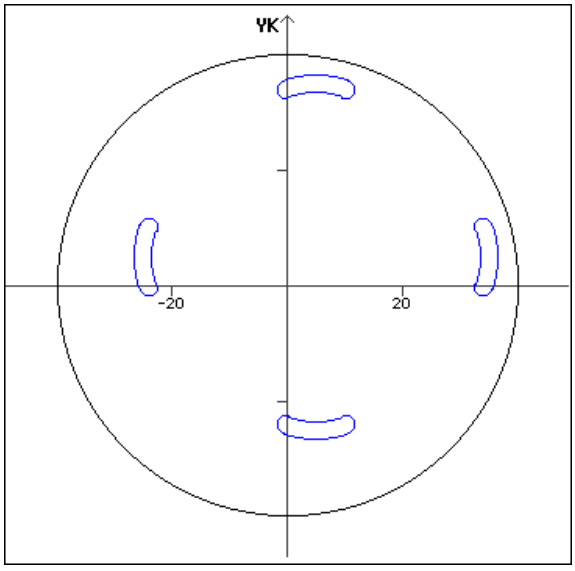
Center of curvature as reference and normal position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the pattern radius plus curvature radius about the pattern center.

Example: Center of curvature as reference, normal position



`N.. G402 Q4 K30 A0 XK5 YK5 H0`	Circular pattern, normal position
`N.. G303 I0 J0 R15 A-20 W20 B3 P1`	Circular slot

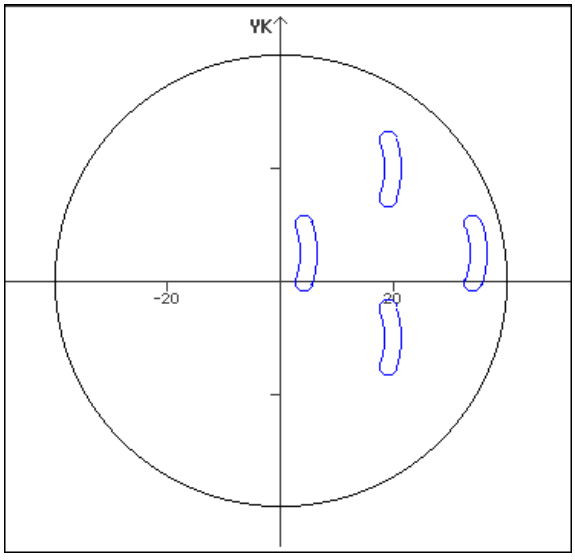
Center of curvature as reference and original position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange the slots at the distance of the pattern radius plus curvature radius about the pattern center while keeping the starting and ending angle.

Example: Center of curvature as reference and original position



`N.. G402 Q4 K30 A0 XK5 YK5 H1`	Circular pattern, original position
`N.. G303 I0 J0 R15 A-20 W20 B3 P1`	Circular slot

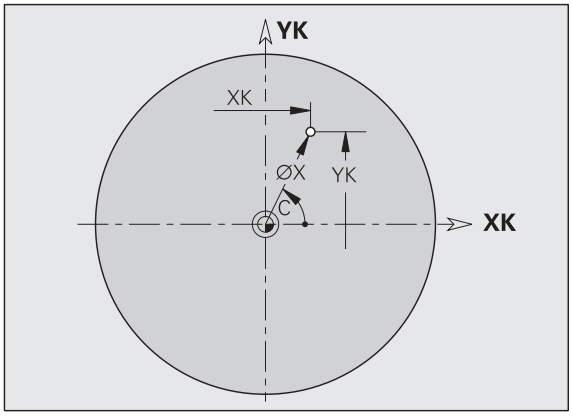
4.10 Front and Rear Face Contours

Starting point of front/rear face contour G100-Geo

G100 defines the starting point of a front or rear face contour.

Parameters

X	Starting point in polar coordinates (diameter)
C	Starting point in polar coordinates (angular dimension)
XK	Starting point in Cartesian coordinates
YK	Starting point in Cartesian coordinates



Line segment in front/rear face contour G101-Geo

G101 defines a line segment in a contour on the front face/rear face.

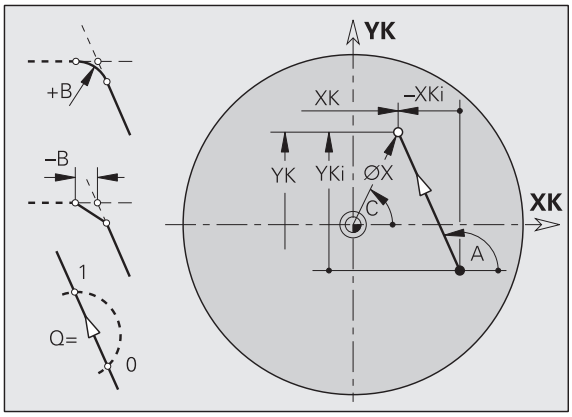
Parameters

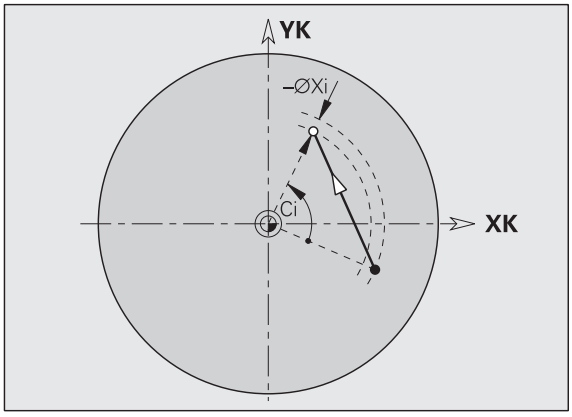
X	End point in polar coordinates (diameter)
C	End point in polar coordinates (angular dimension)
XK	End point in Cartesian coordinates
YK	End point in Cartesian coordinates
A	Angle to positive XK axis
B	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

Q	Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection





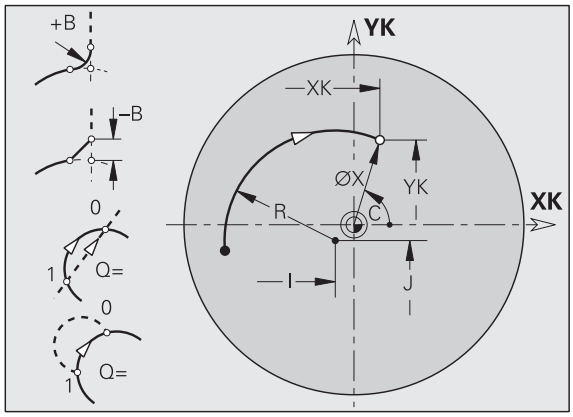
Programming

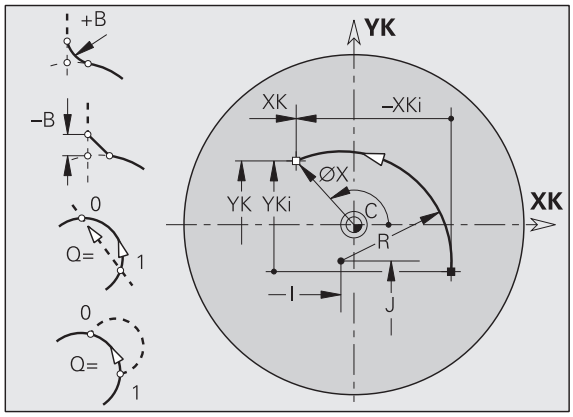
- **X, XK, YX:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal

Circular arc in front/rear face contour G102/ G103-Geo

G102/G103 defines a circular arc in a front or rear face contour. Direction of rotation (see help graphic):

- G102: In clockwise direction
- G102: In counterclockwise direction





Parameters

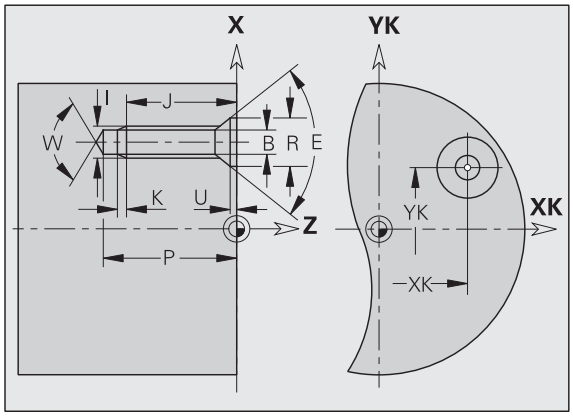
X	End point in polar coordinates (diameter)
C	End point in polar coordinates (angular dimension)
XK	End point in Cartesian coordinates
YK	End point in Cartesian coordinates
R	Radius
I	Center in Cartesian coordinates
J	Center in Cartesian coordinates
B	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point. ■ No entry: Tangential transition ■ B=0: No tangential transition ■ B>0: Rounding radius ■ B<0: Chamfer width
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0): ■ Q=0: Near point of intersection ■ Q=1: Far point of intersection

Programming

- **X, XK, YX:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **I, J:** Absolute or incremental
- End point must not be the starting point (no full circle).

Bore hole on front/rear face G300-Geo

G300 defines a hole with countersinking and thread in a front or rear face contour.



Parameters

XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
B	Hole diameter
P	Depth of hole (excluding point)
W	Point angle (default: 180°)
R	Sinking diameter
U	Sinking depth
E	Sinking angle
I	Thread diameter
J	Thread depth
K	Thread runout length
F	Thread pitch
V	Left-hand or right-hand thread (default: 0)

- V=0: Right-hand thread
- V=1: Left-hand thread

A	Angle to Z axis; angle of the hole

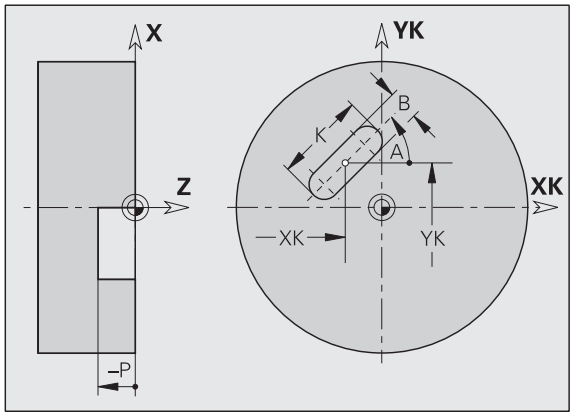
- Range for front face: –90° < A < 90° (default: 0°)
- Range for rear side: 90° < A < 270° (default: 180°)

O	Centering diameter

Machine the G300 holes with G71...G74.

Linear slot on front/rear face G301-Geo

G301 defines a linear slot in a contour on the front face/rear face.



Parameters

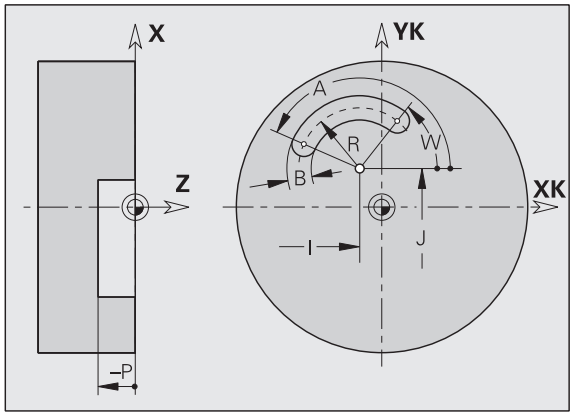
XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
A	Angle to XK axis (default: 0°)
K	Slot length
B	Slot width
P	Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Circular slot on front/rear face G302/G303-Geo

G302/G303 defines a circular slot in a contour on the front face/rear face.

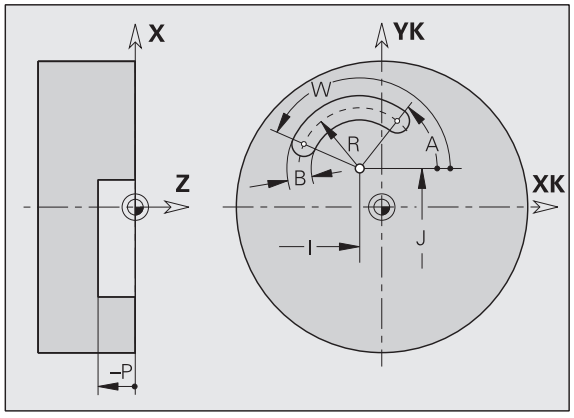
- G302: Circular slot clockwise
- G303: Circular slot counterclockwise



Parameters

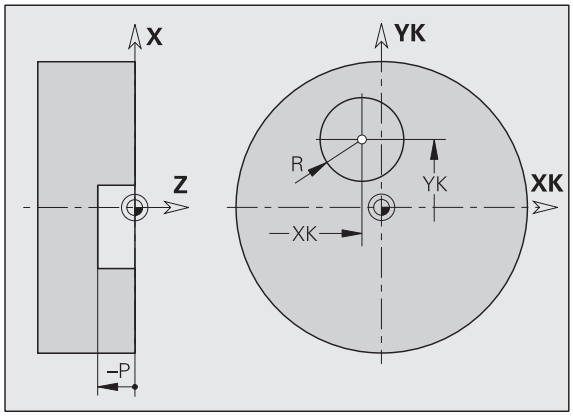
I	Center of curvature in Cartesian coordinates
J	Center of curvature in Cartesian coordinates
R	Curvature radius (reference: center point path of the slot)
A	Starting angle; reference: XK axis (default: 0°)
W	Ending angle; reference: XK axis (default: 0°)
B	Slot width
P	Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island



Full circle on front/rear face G304-Geo

G304 defines a full circle in a contour on the front face/rear face.



Parameters

XK Center in Cartesian coordinates

YK Center in Cartesian coordinates

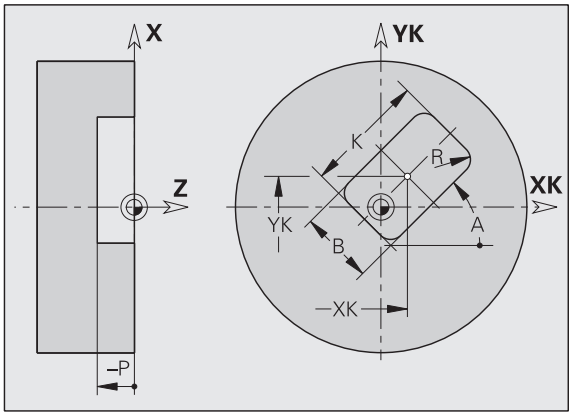
R Radius

P Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Rectangle on front/rear face G305-Geo

G305 defines a rectangle in a contour on the front face/rear face.



Parameters

XK Center in Cartesian coordinates

YK Center in Cartesian coordinates

A Angle to XK axis (default: 0°)

K Length

B (Height) width

R Chamfer/rounding arc (default: 0°)

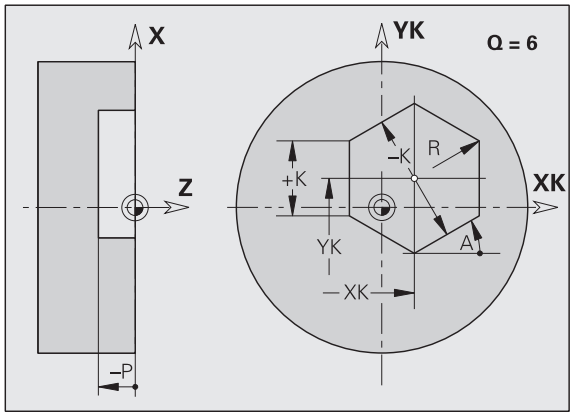
- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Eccentric polygon on front/rear face G307-Geo

G307 defines a polygon in a contour on the front face/rear face.



Parameters

XK Center in Cartesian coordinates

YK Center in Cartesian coordinates

Q Number of edges (Q > 2)

A Angle of a polygon edge to XK axis (default: 0°)

K Edge length

- K>0: Edge length
- K<0: Inscribed circle diameter

B (Height) width

R Chamfer/rounding arc (default: 0°)

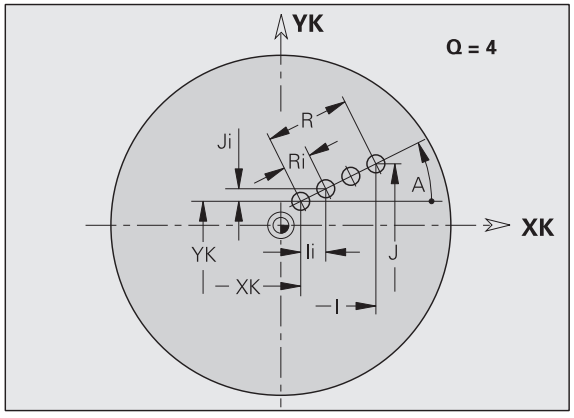
- R>0: Radius of rounding arc
- R<0: Chamfer width

P Depth/height (default: "P" from G308)

- P<0: Pocket
- P>0: Island

Linear pattern on front/rear face G401-Geo

G401 defines a linear hole pattern or figure pattern on the front or rear face. G401 is effective for the hole/figure defined in the following block (G300 to 305, G307).



Parameters

Q Number of figures (default: 1)

XK Starting point in Cartesian coordinates

YK Starting point in Cartesian coordinates

I End point in Cartesian coordinates

J End point in Cartesian coordinates

A Angle of longitudinal axis to XK axis (default: 0°)

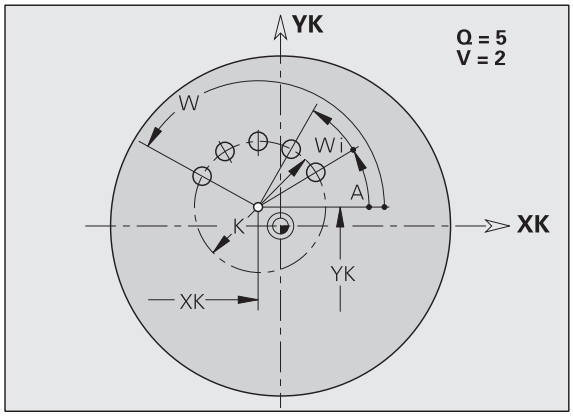
R Total length of pattern

Ri Distance between figures (pattern distance)

- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

Circular pattern on front/rear face G402-Geo

G402 defines a circular hole pattern or figure pattern on the front or rear face. G402 is effective for the hole/figure defined in the following block (G300 to 305, G307).



Parameters

Q Number of figures

K Pattern diameter

A Starting angle – position of the first figure; reference: XK axis; (default: 0°)

W End angle – position of the last figure; reference: XK axis; (default: 360°)

Wi Angle between figures

V Direction – orientation (default: 0)

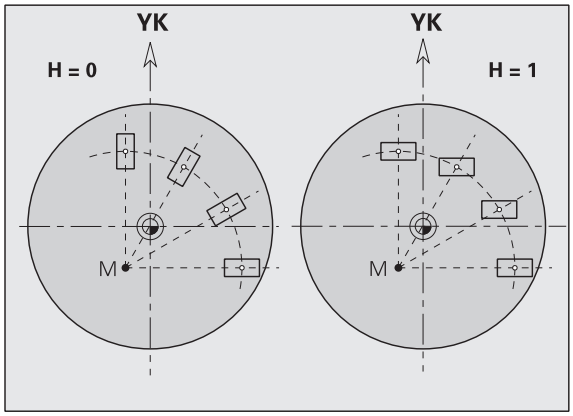
- V=0, without W: Figures are arranged on a full circle
- V=0, with W: Figures are arranged on the longer circular arc
- V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- V=1, with W: Clockwise
- V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- V=2, with W: Counterclockwise
- V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

XK Center in Cartesian coordinates

YK Center in Cartesian coordinates

H Position of the figures (default: 0)

- H=0: Normal position; the figures are rotated about the circle center (rotation)
- H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)

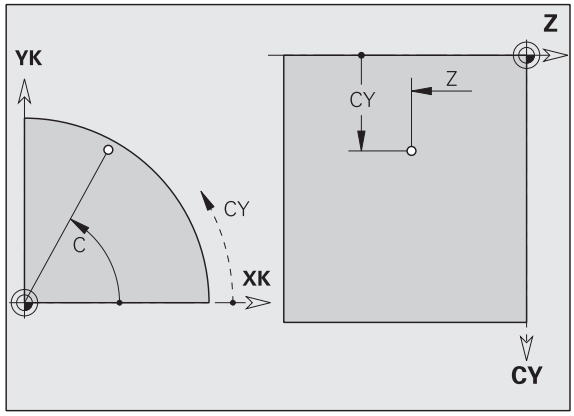


- Program the hole/figure in the following block without a center. Exception: **circular slot**: see "Circular pattern with circular slots" on page 169.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

4.11 Lateral Surface Contours

Starting point of lateral surface contour G110-Geo

G110 defines the starting point of a lateral-surface contour.



Parameters

Z Starting point

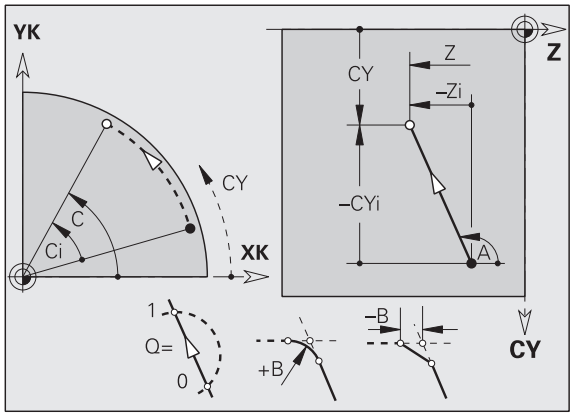
C Starting point (starting angle)

CY Starting point as linear value; reference: unrolled reference diameter

Program either Z, C or Z, CY.

Line segment in a lateral surface contour G111-Geo

G111 defines a line segment in a lateral-surface contour.



Parameters

Z End point

C End point (end angle)

CY End point as linear value; reference: unrolled reference diameter

A Angle to Z axis

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

Q Point of intersection. End point if the line segment intersects a line (default: 0):

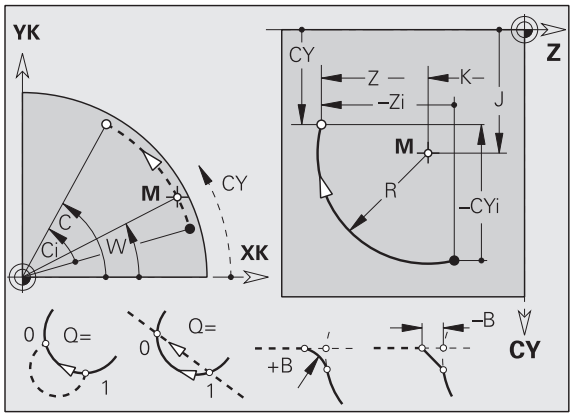
- Q=0: Near point of intersection
- Q=1: Far point of intersection

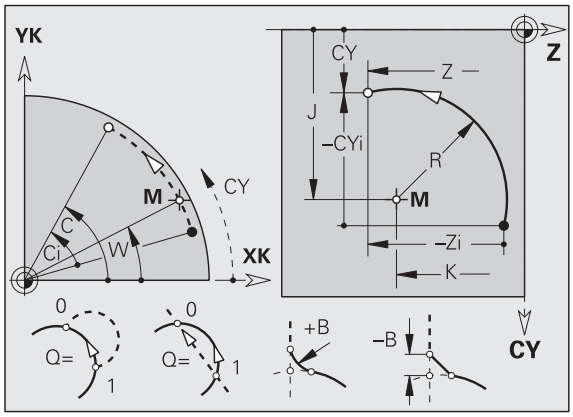
Programming

- **Z, CY:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- Program either Z–C or Z–CY

Circular arc in lateral surface contour G112-/ G113-Geo

G112/G113 defines a circular arc in a lateral-surface contour. Direction of rotation: See help graphic





Parameters

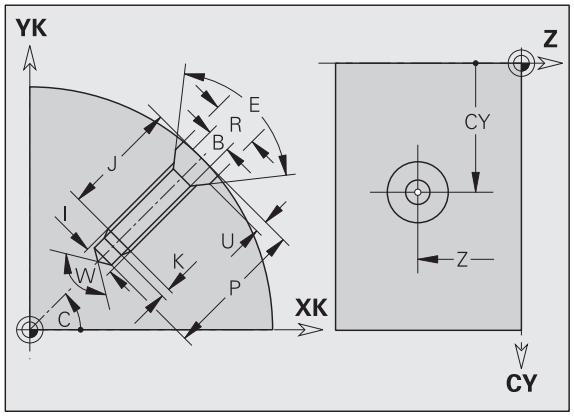
Z	End point
C	End point (end angle)
CY	End point as linear value; reference: unrolled reference diameter
R	Radius
K	Center point in Z direction
W	Angle of the center point
J	Angle of the center point as a linear value
B	Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point. ■ No entry: Tangential transition ■ B=0: No tangential transition ■ B>0: Rounding radius ■ B<0: Chamfer width
Q	Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0): ■ Q=0: Near point of intersection ■ Q=1: Far point of intersection

Programming

- **Z, CY:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **K, J:** Absolute or incremental
- Program either Z–C or Z–CY, and either K–W or K–J
- Program either center or radius
- For radius: Only arcs <= 180° are possible

Hole on lateral surface G310-Geo

G310 defines a hole with countersink and thread in a lateral surface contour.



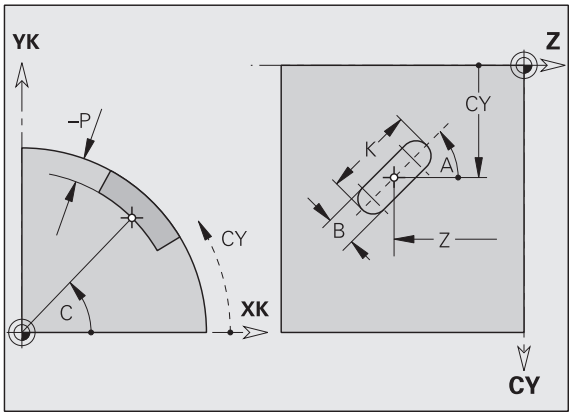
Parameters

Z	Center (Z position)
C	Center (angle)
CY	Center as linear value; reference: unrolled reference diameter
B	Diameter of hole
P	Depth of hole (excluding point)
W	Point angle (default: 180°)
R	Sinking diameter
U	Sinking depth
E	Sinking angle
I	Thread diameter
J	Thread depth
K	Thread runout length
F	Thread pitch
V	Left-hand or right-hand thread (default: 0)
	■ V=0: Right-hand thread
	■ V=1: Left-hand thread
A	Angle to Z axis; range: 0° < A < 180°; (default: 90° = vertical hole)
O	Centering diameter

Machine the G310 holes with G71...G74.

Linear slot on lateral surface G311-Geo

G311 defines a linear slot in a lateral-surface contour.



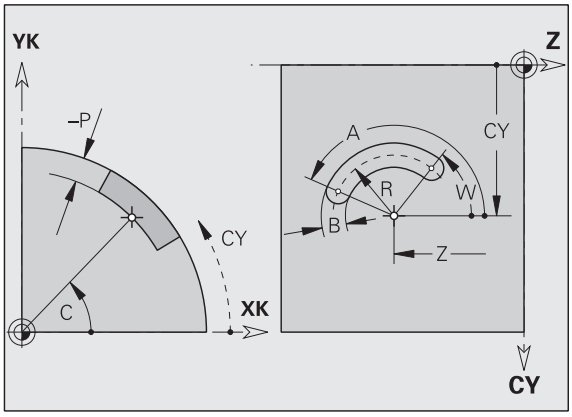
Parameters

Z	Center (Z position)
C	Center (angle)
CY	Center as linear value; reference: unrolled reference diameter
A	Angle to Z axis (default: 0°)
K	Slot length
B	Slot width
P	Pocket depth (default: “P” from G308)

Circular slot on lateral surface G312/G313-Geo

G312/G313 defines a circular slot in a lateral-surface contour

- G312: Circular slot clockwise
- G313: Circular slot counterclockwise

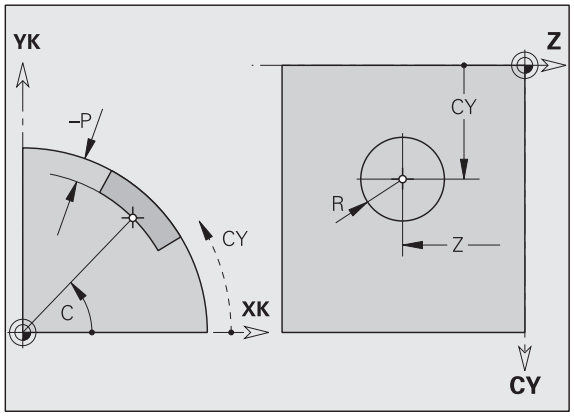


Parameters

Z	Center
C	Center (angle)
CY	Center as linear value; reference: unrolled reference diameter
R	Radius; reference: center point path of the slot
A	Starting angle; reference: Z axis (default: 0°)
W	End angle; reference: Z axis
B	Slot width
P	Pocket depth (default: “P” from G308)

Full circle on lateral surface G314-Geo

G314 defines a full circle in a lateral-surface contour.



Parameters

Z Center

C Center (angle)

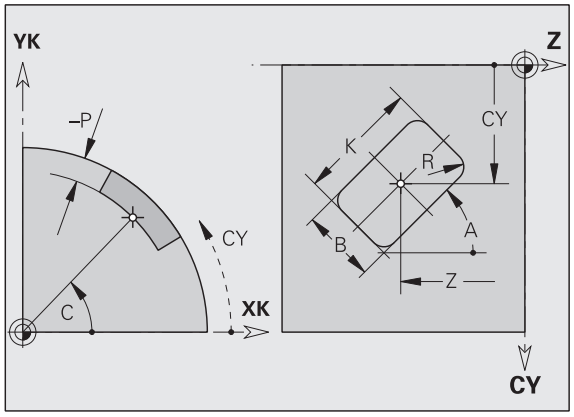
CY Center as linear value; reference: unrolled reference diameter

R Radius

P Pocket depth (default: "P" from G308)

Rectangle on lateral surface G315-Geo

G315 defines a rectangle in a lateral-surface contour.



Parameters

Z Center

C Center (angle)

CY Center as linear value; reference: unrolled reference diameter

A Angle to Z axis (default: 0°)

K Length

B Width

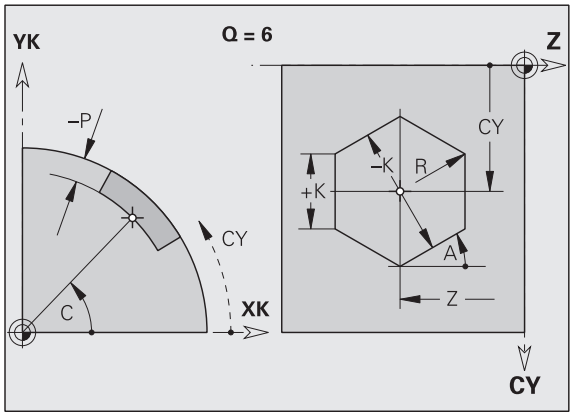
R Chamfer/rounding arc (default: 0°)

- R>0: Radius of rounding arc
- R<0: Chamfer width

P Pocket depth (default: "P" from G308)

Eccentric polygon on lateral surface G317-Geo

G317 defines a polygon in a lateral-surface contour.



Parameters

Z Center

C Center (angle)

CY Center as linear value; reference: unrolled reference diameter

Q Number of edges (Q > 2)

A Angle to Z axis (default: 0°)

K Edge length

- K>0: Edge length
- K<0: Inside diameter of circle

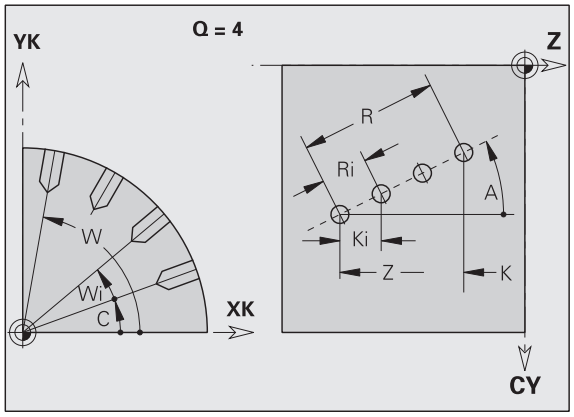
R Chamfer/rounding arc (default: 0°)

- R>0: Radius of rounding arc
- R<0: Chamfer width

P Pocket depth (default: “P” from G308)

Linear pattern on lateral surface G411-Geo

G411 defines a linear hole or figure pattern on the lateral surface. G411 is effective for the hole/figure defined in the following block (G310.0.315, G317).



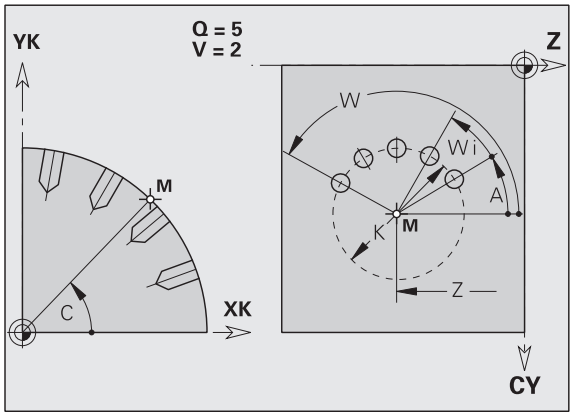
Parameters

Q	Number of figures (default: 1)
Z	Start point
C	Starting point (starting angle)
CY	Starting point as linear value; reference: unrolled reference diameter
K	End point
Ki	Distance between figures in Z direction
W	End point (end angle)
Wi	Angular distance between figures
A	Angle to Z axis; (default: 0°)
R	Total length of pattern
Ri	Distance between figures (pattern distance)

- If you program Q, Z and C, the holes/figures are arranged at a regular spacing on the lateral surface.
- Program the hole/figure in the following block without a center.
- The milling cycle calls the hole/figure in the following block—not the pattern definition.

Circular pattern on lateral surface G412-Geo

G412 defines a circular hole or figure pattern on the lateral surface. G412 is effective for the hole/figure defined in the following block (G310.0.315, G317).



Parameters

Q Number of figures

K Pattern diameter

A Starting angle – position of the first figure; reference: Z axis; (default: 0°)

W End angle – position of the last figure; reference: Z axis (default: 360°)

Wi Angle between figures

V Direction—orientation (default: 0)

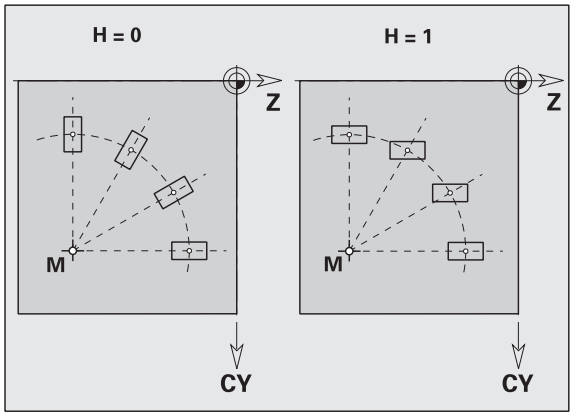
- V=0, without W: Figures are arranged on a full circle
- V=0, with W: Figures are arranged on the longer circular arc
- V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
- V=1, with W: Clockwise
- V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
- V=2, with W: Counterclockwise
- V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)

Z Center of pattern

C Center of pattern (angle)

H Position of the figures (default: 0)

- H=0: Normal position; the figures are rotated about the circle center (rotation)
- H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)



- Program the hole/figure in the following block without a center. Exception: **circular slot**: see "Circular pattern with circular slots" on page 169.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

4.12 Tool Positioning

Rapid traverse G0

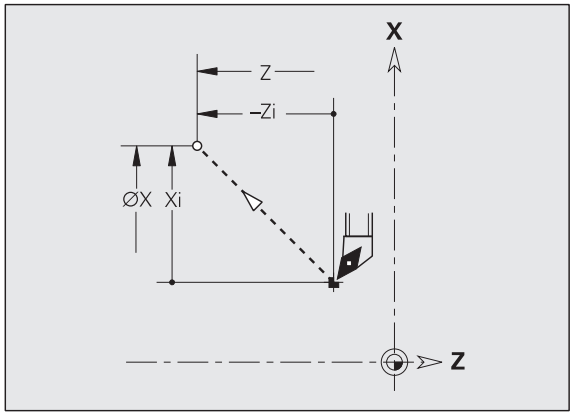
G0 moves at rapid traverse along the shortest path to the target point.

Parameters

X Target point (diameter)

Z Target point

Programming X, Z: Absolute, incremental or modal



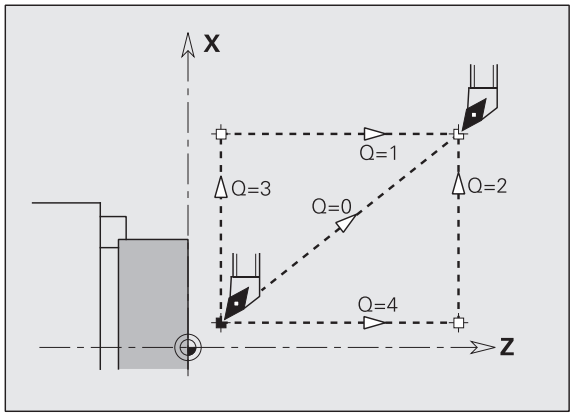
Setting the tool change position G14

G14 moves the slide at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change position.

Parameters

Q Sequence. Determines the course of traverse movements (default: 0)

- Q=0: Diagonal path of traverse
- Q=1: First X, then Z direction
- Q=2: First Z, then X direction
- Q=3: Only X direction, Z remains unchanged
- Q=4: Only Z direction, X remains unchanged



Example: G14

```
. . .
N1 G14 Q0          [Move to the tool change
position]
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
. . .
```

Rapid traverse to machine coordinates G701

G701 moves at rapid traverse along the shortest path to the target point.

Parameters

X End point (diameter)

Z End point

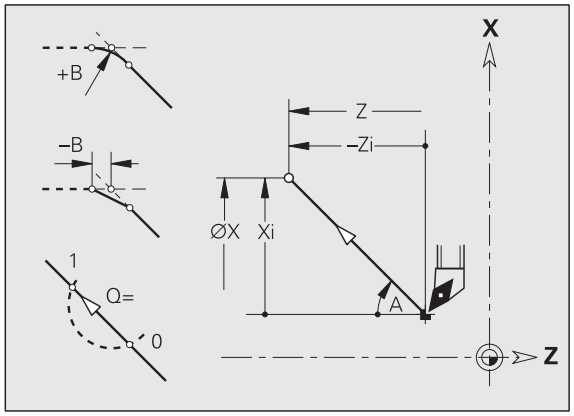


X, Z refer to the machine zero point and the slide zero point.

4.13 Simple Linear and Circular Movements

Linear path G1

G1 moves the tool on a linear path at the feed rate to the "end point."



Parameters

X End point (diameter)

Z End point

A Angle (angular direction: see help graphic)

Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E Special feed factor for chamfer/rounding arc (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)

Programming X, Z: Absolute, incremental, modal or "?"

Circular path G2/ G3

G2/G3 moves the tool in a circular arc at the feed rate to the "end point." The center dimensioning is **incremental.** Direction of rotation (see help graphic):

- G2: In clockwise direction
- G3: In counterclockwise direction

Parameters

X End point (diameter)

Z End point

R Radius (0 < R <= 200 000 mm)

I incremental center point (distance from starting point to center point; radius)

K Incremental center point (distance from starting point to center)

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

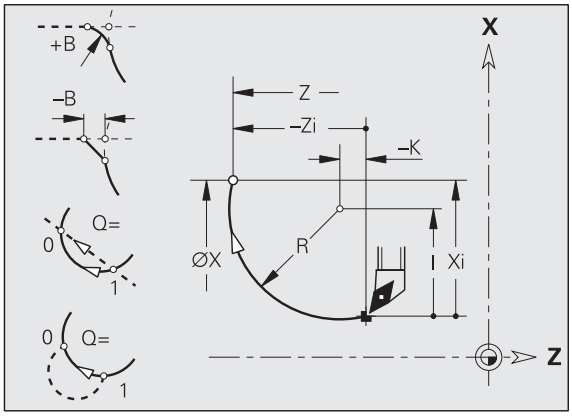
E Special feed factor for chamfer/rounding arc (default: 1)

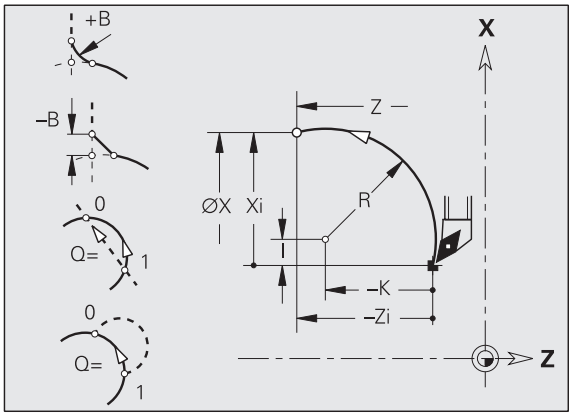
Special feed rate = active feed rate * E (0 < E <= 1)

Programming X, Z: Absolute, incremental, modal or "?"

Danger of collision!

If V variables are used for calculating the address parameters, a limited contour check is carried out. Ensure that the variable values produce a circular arc.





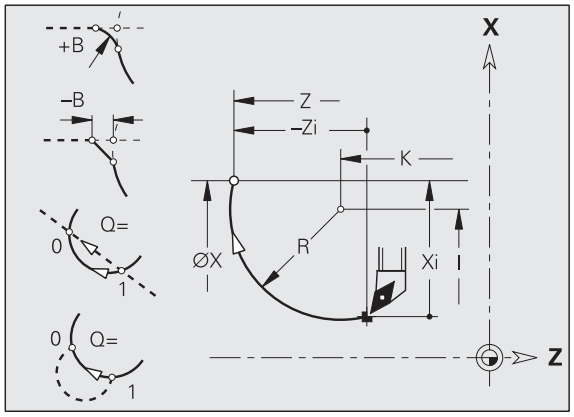
Example: G2, G3

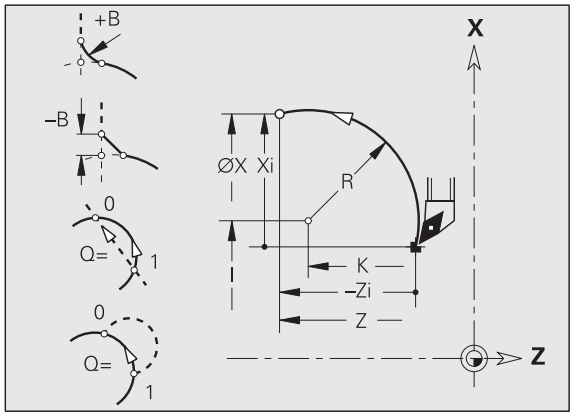
```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X15 B-0.5 E0.05
N6 G1 Z-25 B0
N7 G2 X45 Z-32 R36 B2
N8 G1 A0
N9 G2 X80 Z-80 R20 B5
N10 G1 Z-95 B0
N11 G3 X80 Z-135 R40 B0
N12 G1 Z-140
N13 G1 X82 G40
. . .
```

Circular path G12/ G13

G12/G13 moves the tool in a circular arc at the feed rate to the "end point." The center dimensioning is **absolute.** Direction of rotation (see help graphic):

- G12: In clockwise direction
- G13: In counterclockwise direction





Parameters

X End point (diameter)

Z End point

R Radius (0 < R <= 200 000 mm)

I Absolute center point (radius)

K Absolute center point

Q Point of intersection. End point if the circular arc intersects a line segment or another circular arc (default: 0):

- Q=0: Near point of intersection
- Q=1: Far point of intersection

B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.

- No entry: Tangential transition
- B=0: No tangential transition
- B>0: Rounding radius
- B<0: Chamfer width

E Special feed factor for chamfer/rounding arc (default: 1)

Special feed rate = active feed rate * E (0 < E <= 1)

Programming X, Z: Absolute, incremental, modal or "?"

Danger of collision!

If V variables are used for calculating the address parameters, a limited contour check is carried out. Ensure that the variable values produce a circular arc.

4.14 Feed Rate and Spindle Speed

Rotational speed limiting G26

G26: Main spindle; Gx26: Spindle x (x: 1...3)

The speed limitation remains in effect until the end of the program or until a new value is programmed for G26/Gx26.

Parameters

S (Maximum) speed



Actual S > "absolute maximum speed" (MP 805, ff), applies to the parameter value.

Example: G26

```
. . .
N1 G14 Q0
N1 G26 S2000                    [maximum speed]
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
. . .
```

Acceleration (slope) G48

G48 defines acceleration, deceleration, and the maximum feed rate. G48 is a modal function.

Without G48, these parameters apply:

- Approach acceleration, braking acceleration: MP 1105, ... Acceleration/braking of linear axis
- Maximum feed rate: MP 1101, ... Maximum axis velocity

Parameters

E Acceleration starting an axis (default: parameter value)

F Acceleration for braking an axis (default: parameter value)

H Programmed acceleration on/off

- H=0: Switch off programmed acceleration after next traverse path
- H=1: Switch on programmed acceleration

P Maximum feed rate (default: parameter value)



- If P > parameter value, the parameter value applies.
- E, F and P refer to the X or Z axis. The acceleration/feed-rate of the slide is higher for non-paraxial paths of traverse.

Interrupted feed G64

G64 interrupts the programmed feed for a short period of time. G64 is a modal function.

Parameters

E Pause duration (0.01 s < E < 99.99 s)

F Feed duration (0.01 s < E < 99.99 s)

- For switch-on, program G64 with E and F.
- For switch-off, program G64 without parameters.

Example: G64

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G64 E0.1 F1          [interrupted feed on]
N3 G0 X0 Z2
N4 G42
N5 G1 Z0
N6 G1 X20 B-0.5
N7 G1 Z-12
N8 G1 Z-24 A20
N9 G1 X48 B6
N10 G1 Z-52 B8
N11 G1 X80 B4 E0.08
N12 G1 Z-60
N13 G1 X82 G40
N14 G64                 [interrupted feed off]
. . .
```

Feed per minute for rotary axes G192

G192 defines the feed rate when only a rotary axis (auxiliary axis) is moved.

Parameters

F Feed in degrees/minute.

Feed per tooth Gx93

Gx93 (x: spindle 1...3) defines the **drive-dependent** feed rate with respect to the number of teeth of the cutter.

Parameters

F Feed per tooth in mm/tooth or inch/tooth



The actual value display shows the feed rate in mm/rev.

Example: G193

```
. . .
N1 M5
N2 T1 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30
N5 G110 C0
N6 G0 X122 Z-50
N7 G...
N8 G...
N9 M15
. . .
```

Constant feed rate G94 (feed per minute)

G94 defines the feed rate **independent of drive.**

Parameters

F Feed per minute in mm/min or in./min

Example: G94

```
. . .
N1 G14 Q0
N2 T3 G94 F2000 G97 S1000 M3
N3 G0 X100 Z2
N4 G1 Z-50
. . .
```

Feed per revolution Gx95

G95: Main spindle; Gx95: Spindle x (x: 1...3)

Gx95 defines a **drive-dependent** feed rate.

Parameters

F Feed rate in mm/revolution or inch/revolution

Example: G95, Gx95

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

Constant surface speed Gx96

G96: Main spindle; Gx96: Spindle x (x: 1...3)

The spindle speed is dependent on the X position of the tool tip or on the diameter of the driven tools.

Parameters

S Cutting speed in m/min or ft/min

Example: G96, G196

```
. . .
N1 T3 G195 F0.25 G196 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6 G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
. . .
```

Speed Gx97

G97: Main spindle; Gx97: Spindle x (x: 1...3)

Constant spindle speed.

Parameters

S Speed in revolutions per minute



G26/Gx26 limits the spindle speed.

Example: G97, G197

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G97 S1000 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

4.15 Tool-Tip and Cutter Radius Compensation

Tool-tip radius compensation (TRC)

If TRC is not used, the theoretical tool tip is the reference point for the paths of traverse. This might lead to inaccuracies when the tool moves along non-paraxial paths of traverse. The TRC function corrects programmed paths of traverse.

The TRC (Q=0) **reduces** the feed rate for circular arcs if the shifted radius < the original radius. The TRC corrects the special feed rate when a rounding arc is machined as transition to the next contour element.

Reduced feed rate = feed rate * (shifted radius / original radius)

Milling cutter radius compensation (MCRC)

When the MCRC function is not active, the system defines the center of the cutter as the zero point for the paths of traverse. With the CNC PILOT accounts for the outside cutting radius when moving along the programmed paths of traverse. The **recessing, roughing and milling cycles** already include TRC/MCRC calls. The TRC/MCRC must be switched off when these cycles are called.



- If the tool radii are > than the contour radii, the TRC/MCRC might cause endless loops. **Recommendation:** Use the finishing cycle G890 or milling cycle G840.
- Never program the MCRC during a perpendicular approach to the machining plane.
- Note on calling subprograms: Switch the TRC/MCRC off
  - in the subprogram in which it was switched on.
  - in the main program if it was switched on there.

G40: Switch off TRC/MCRC

G40 is used to deactivate TRC/MCRC. Please note:

- The TRC/MCRC remains in effect until a block with G40 is reached.
- The block containing G40, or the block after G40 only permits a linear path of traverse (G14 is not permissible).

Function of the TRC/MCRC

`. . .`	
`N.. G0 X10 Z10`	
`N.. G41 G0 Z20`	Path of traverse: from X10/Z10 to X10+TRC/ Z20+TRC
`N.. G1 X20`	The path of traverse is “shifted” by the TRC
`N.. G40 G0 X30 Z30`	Path of traverse from X20+TRC/Z20+TRC to X30/ Z30
`. . .`	

G41/G42: Switch on TRC/MCRC

G41: Switch on TRC/MCRC—compensation of the tool-tip/cutter radius to the **left** of the contour in traverse direction.

G42: Switch on TRC/MCRC—compensation of the tool-tip/cutter radius to the **right** of the contour in traverse direction.

Parameters

Q Plane (default: 0)

- Q=0: TRC on the turning plane (XZ plane)
- Q=1: MCRC on the front face (XC plane)
- Q=2: MCRC on the lateral surface (ZC plane)
- Q=3: MCRC on the front face (XY plane)
- Q=4: MCRC on the lateral surface (YZ plane)

H Output (only with TRC) – (default: 0)

- H=0: Intersecting areas which are programmed in directly successive contour elements are not machined.
- H=1: The complete contour is machined—even if certain areas are intersecting.

O Feed rate reduction (default: 0)

- O=0: Feed rate reduction active
- O=1: No feed rate reduction

Please note:

- Program a straight line segment (G0/G1) in or after the block containing G41/G42.
- The TRC/MCRC is taken into account from the next path of traverse.

Example: G40, G41, G42

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42          [TRC on, to the right of the
contour]
N4 G1 Z0
N5 G1 X20 B-0.5
N6 G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40                  [TRC off]
. . .
```

4.16 Zero Point Shifts

You can program several zero point shifts in one NC program. The relationships of the coordinates (for blank/finished part, auxiliary contours) are retained by the zero offset description.

G920 temporarily deactivates zero point shifts—G980 reactivates them.

Overview of zero point shifts	
G51:	Page 199
■ Relative shift ■ Programmed shift ■ Reference: Previously defined workpiece zero point	
G53, G54, G55:	Page 199
■ Relative shift ■ Shift from parameters ■ Reference: Previously defined workpiece zero point	
G56:	Page 200
■ Additive shift ■ Programmed shift ■ Reference: Workpiece zero point defined at present	
G59:	Page 201
■ Absolute shift ■ Programmed shift ■ Reference: Machine zero point	

Zero point shift G51

G51 shifts the workpiece zero point by Z (or X). The shift is referenced to the workpiece zero point defined in setup mode.

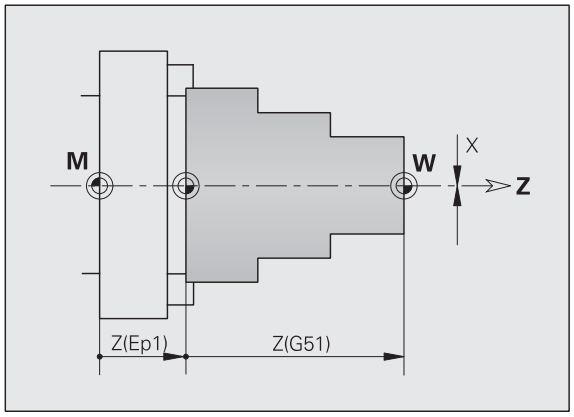
Parameters

X Shift (radius)

Z Shift

Even if you shift the zero point several times with G51, it is always referenced to the workpiece zero point defined in setup mode.

The zero point shift is valid until program end, or until it is canceled by other zero point shifts.



Example: G51

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X62 Z5
N3 G810 NS7 NE12 P5 I0.5 K0.2
N4 G51 Z-28                [zero point shift]
N5 G0 X62 Z-15
N6 G810 NS7 NE12 P5 I0.5 K0.2
N7 G51 Z-56                [zero point shift]
. . .
```

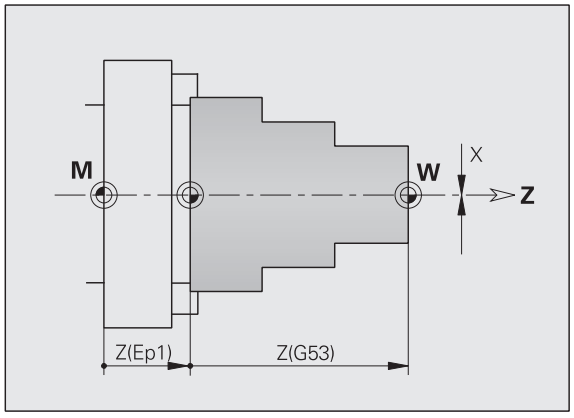
Parameter-dependent zero offset G53, G54, G55

G53 to G55 shifts the workpiece zero point by the value defined in the setup parameters 3, 4, 5. The shift is referenced to the workpiece zero point defined in setup mode.

Even if you program G53, G54, G55 more than once, the reference point remains the workpiece zero point defined in setup mode.

The zero point shift is valid until program end or until it is canceled by other zero point shifts.

A shift in X is entered as a radius.



Additive zero point shift G56

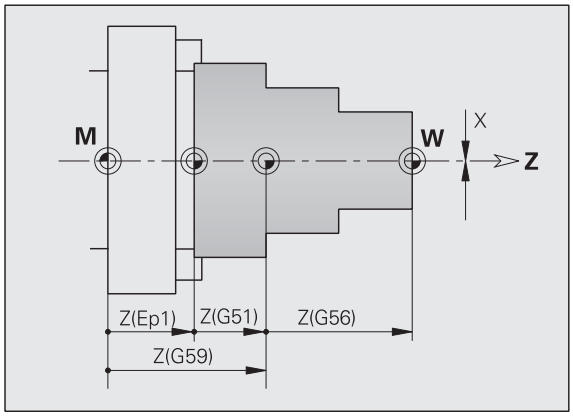
G56 shifts the workpiece zero point by Z (or X). The shift is referenced to the currently active workpiece zero point.

Parameters

X Shift (radius value) – (default: 0)

Z Shift

If you shift the workpiece zero point more than once with G56, the shift is always added to the currently active zero point.



Example: G56

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X62 Z5
N3 G810 NS7 NE12 P5 I0.5 K0.2
N4 G56 Z-28                [zero point shift]
N5 G0 X62 Z5
N6 G810 NS7 NE12 P5 I0.5 K0.2
N7 G56 Z-28                [zero point shift]
. . .
```

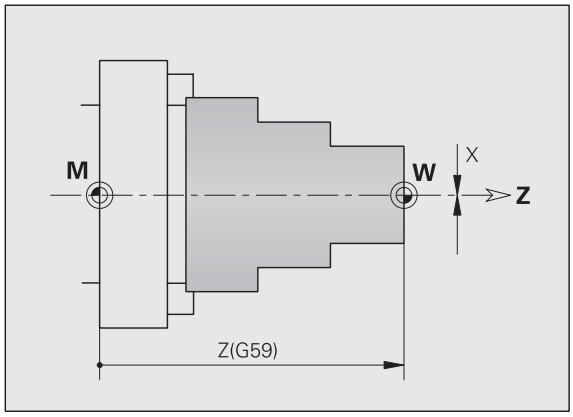
Absolute zero point shift G59

G59 sets the workpiece zero point to X, Z. The new zero point remains in effect to the end of the program.

Parameters

X Shift (radius)

Z Shift



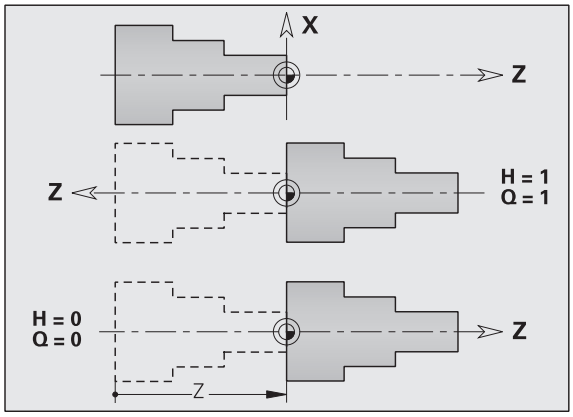
G59 cancels all previous zero point shifts (with G51, G56 or G59).

Example: G59

```
. . .
N1 G59 Z256                [zero point shift]
N2 G14 Q0
N3 T3 G95 F0.25 G96 S200 M3
N4 G0 X62 Z2
. . .
```

Mirror/shift contour G121

G121 mirrors and/or shifts the blank and finished part contours. The contour is mirrored at the X axis and shifted in Z direction. The workpiece zero point is not affected.



Parameters

H Type of transformation (default: 0)

- H=0: Contour shift, not mirroring
- H=1: Contour shift, mirroring and reversing the direction of the contour description

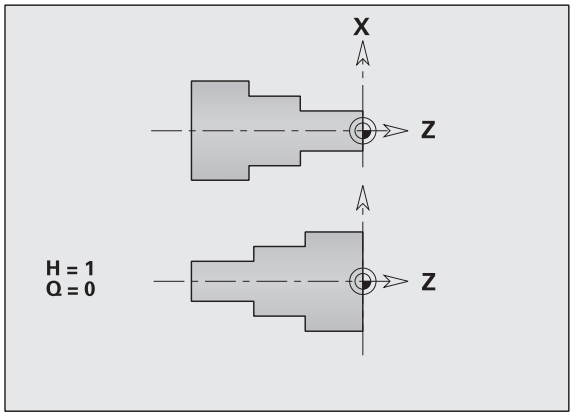
Q Mirroring the Z axis of the coordinate system (default: 0)

- Q=0: Do not mirror
- Q=1: Mirror

Z Shift. Shift coordinate system in Z direction (default: 0)

D Mirror XC/XCR (mirroring and shifting front and rear face contours) – (default: 0)

- Q=0: Do not mirror/shift
- Q=1: Mirror/shift



G121 allows you to use the blank and finished part descriptions for front and rear-face machining.

- Cylindrical surface contours are mirrored/shifted like turning contours.
- Auxiliary contours are not mirrored.
- Note: Q=1 mirrors the coordinate system and the contour; H=1 mirrors only the contour.

Contour shift, mirroring the coordinate system

`N.. . . .`	Rear side machining on the opposing spindle
`N.. G121 H1 Q1 Z.. D1`	Shifts and mirrors the contour; mirrors the coordinate system.
`N.. . . .`	

Shift contour, do not mirror

`N.. . . .`	Rear side machining on the opposing spindle
`N.. G121 H0 Q0 Z.. D1`	Shifts the contour
`N.. . . .`	

Contour mirroring and shifting

`N.. . . .`	Rear-face machining with one spindle (manual rechucking)
`N.. G121 H1 Q0 Z.. D1`	Shifts and mirrors the contour
`N.. . . .`	

4.17 Oversizes

Switch off oversize G50

G50 switches off oversizes defined with G52/G39 Geo for the following cycle. Program G50 before the cycle.

To ensure compatibility the G52 code is also supported for switching off the oversizes. HEIDENHAIN recommends using G50 for new NC programs.

Axis-parallel oversize G57

G57 defines different oversizes for X and Z. Program G57 before the cycle call.

Parameters

X Oversize X (diameter value) – only positive values

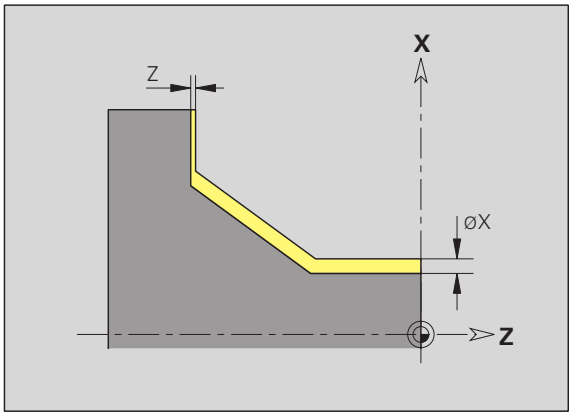
Z Oversize Z – only positive values

G57 is effective in the following cycles. After cycle run, the oversizes are

- Deleted: G810, G820, G830, G835, G860, G869, G890
- **Not** deleted: G81, G82, G83



If the oversizes are programmed with G57 **and** in the cycle itself, the cycle oversizes apply.



Example: G57

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G57 X0.2 Z0.5          [paraxial oversize]
N4 G810 NS7 NE12 P5
. . .
```

Contour-parallel oversize (equidistant) G58

G58 defines an equidistant oversize. Program G58 before the cycle call. A negative oversize during finishing is permitted with G890.

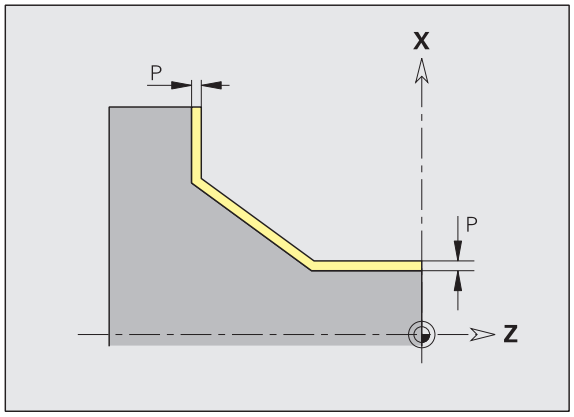
Parameters

P Oversize

G58 is effective in the following cycles. After cycle run, the oversizes are

- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G83

If an oversize is programmed with G58 **and** in the cycle, the oversize from the cycle is used.



Example: G58

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G58 P2          [contour-parallel oversize]
N4 G810 NS7 NE12 P5
. . .
```

4.18 Safety Clearances

Safety clearance G47

G47 defines the safety clearance for

- the turning cycles: G810, G820, G830, G835, G860, G869, G890.
- the drilling cycles G71, G72, G74.
- the milling cycles G840...G846.

Parameters

P Safety clearance

G47 without parameters activates the parameter values (machining parameters 2, ... – safety clearances).



G47 replaces the safety clearance set in the machining parameters or that set in G147.

Safety clearance G147

G147 defines the safety clearance for

- the milling cycles G840...G846.
- the drilling cycles G71, G72, G74.

Parameters

I Safety clearance to the milling plane (only for milling operations)

K Safety clearance in approach direction (feed)

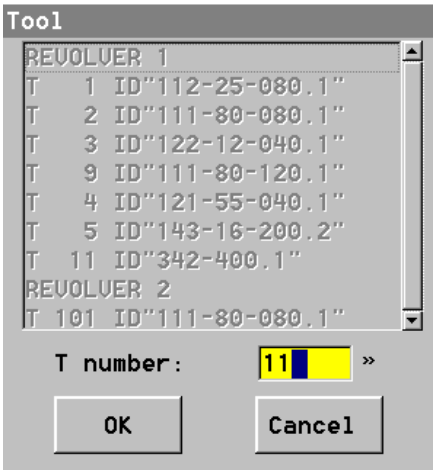


G147 replaces the safety clearance set in the parameters (machining parameters 2, ...) or that set in G47.

4.19 Tools, Types of Compensation

Tool call – T

The CNC PILOT displays the tool assignment defined in the TURRET section. You can enter the T number directly or select it from the tool list (switch with the CONTINUE soft key).



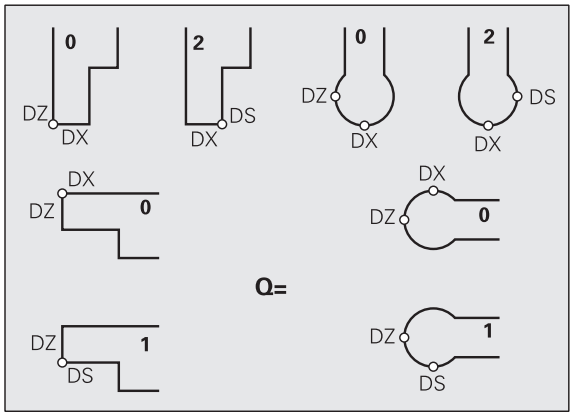
(Changing the) tool edge compensation G148

G148 defines the values compensating for wear. DX, DZ become effective after program start and after a T command.

Parameters

Q Selection (default: 0)

- O=0: DX, DZ active—DS inactive
- O=1: DS, DZ active—DX inactive
- O=2: DX, DS active—DZ inactive



The recessing cycles G860, G866, G869 automatically take the "correct" wear compensation into account.

Example: G148

```
. . .
N1 T3 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7 G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151                [recessing finishing]
N11 G148 O0             [change compensation]
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 O2
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
. . .
```

Additive compensation G149

The CNC PILOT manages 16 tool-independent compensation values. One G149 followed by a D number activates the additive compensation function. G149 D900 deactivates the additive compensation function.

Parameters

D Additive compensation (default: D900):

- D900: deactivates the additive compensation
- D901 to D916: activates the additive compensation

Programming:

- The compensation becomes effective after the tool has moved in the compensation direction by the compensation value. Therefore, program G149 one block before the block containing the path of traverse to which the compensation is to apply.
- Additive compensation remains in effect until:
  - To the next G149 D900
  - To the next tool change
  - End of program

Example: G149

```
. . .
N1 T3 G96 S200 G95 F0.4 M4
N2 G0 X62 Z2
N3 G89
N4 G42
N5 G0 X27 Z0
N6 G1 X30 Z-1.5
N7 G1 Z-25
N8 G149 D901          [activate compensation]
N9 G1 X40 B-1
N10 G1 Z-50
N11 G149 D902
N12 G1 X50 B-1
N13 G1 Z-75
N14 G149 D900      [deactivate compensation]
N15 G1 X60 B-1
N16 G1 Z-80
N17 G1 X62
N18 G80
. . .
```

Compensation of right-hand tool tip G150
Compensation of left-hand tool tip G151

G150/G151 defines the tool reference point for recessing and button tools.

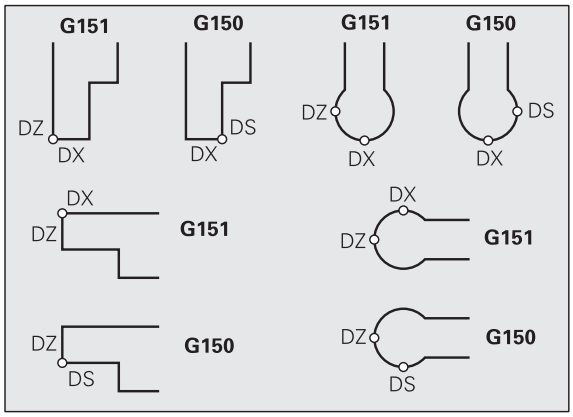
- G150: Reference point is on right tip
- G151: Reference point is on left tip.

G150/G151 is effective from the block in which it is programmed and remains in effect up to

- the next tool change
- program end.



- The displayed actual values always refer to the tool tip defined in the tool data.
- If you use TRC, after G150/G151 you must also adjust G41/G42.



Example: G150, G151

```
. . .
N1 T3 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7 G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151                    [recessing finishing]
N11 G148 00
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 02
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
. . .
```

Adding tool dimensions G710

When a T command is programmed, the CNC PILOT replaces the previous tool dimensions with new tool dimensions. When you activate the adding function with G710 Q1, the dimensions of the new tool are **added** to the dimensions of the previous tool.

Parameters

Q Adding tool dimensions

- Q=0: Off
- Q=1: On

Application example
For full-surface machining, the workpiece is transferred to a rotating gripper after having been machined on the front face. The rear face is machined by stationary tools. To do this, the dimensions of the rotating gripper are added to the dimensions of the stationary tool.

Example of adding tool dimensions

`. . .`	
`TURRET 1`	
`. . .`	
`T14 ID"GRIPPER"`	Rotating gripper
`. . .`	
`REVOLVER 2 [TURRET]`	Stationary tools on tool carrier 2
`T2001 ID"116-80-080.1"`	Roughing tool for rear-face machining
`. . .`	
`BEARBEITUNG [MACHINING]`	
`. . .`	
`N100 T14`	Insert rotating gripper
`N101 L"EXGRIGF" V1`	Transfer workpiece from main spindle to rotating gripper (expert program)
`N102 G710 Q1`	Adding tool dimensions
`N103 T2001`	Add the dimensions of rotating gripper and tool
`. . .`	

4.20 Contour-Based Turning Cycles

Working with cycles

Finding the block references:



Activate the contour view:
- Press the soft key or select the menu item “Graphic.”
- Place cursor in NS or NE input field



Switch to graphic window:
- Press the CONTINUE soft key

Select the contour element:
- Use the horizontal arrow keys to select the contour element
- Use the vertical arrow keys to switch between contours (also face contours, etc.).
- Confirm the block number of the contour element with ENTER

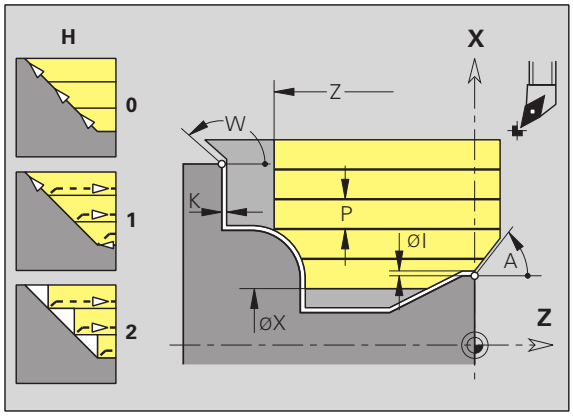
If you press the vertical arrow keys, the CNC PILOT also considers contours that are not displayed on the screen.

Cutting limit

The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Longitudinal roughing G810

G810 machines the contour area from NS to NE defined by NS, NE. If required, the area to be machined is divided into several sections (example: with contour valleys).



Parameters

NS	Starting block number (beginning of contour section)
NE	End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P	Maximum infeed
I	Oversize in X direction (diameter value) – (default: 0)
K	Oversize in Z direction (default: 0)

Parameters

E Approach behavior

- E=0: Descending contours are not machined.
- E>0: Plunging feed rate
- No input: Feed rate reduction depending on the plunge angle – maximum 50%

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

H Type of departure (default: 0)

- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45°; no contour smoothing

A Approach angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

W Departing angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

Q Type of retraction at cycle end (default: 0)

- Q=0: Return to starting point (first X, then Z direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

D Omit elements. The following undercuts, relief turns and recesses are not run (default: 0):

	G22	G23 H0	G23 H1	G25 H4	G25 H5/6	G25 H7 to H9
D=0	•	•	•	•	•	•
D=1	•	•	•	–	–	–
D=2	•	•	–	•	•	•
D=3	•	•	–	–	–	–
D=4	•	•	–	•	•	–

"•": Do not machine the elements

Parameters

B Slide lead with 4-axis machining

- B=0: Both slides work at the same diameter—at double feed rate
- B<>0: Distance to the leading slide (the approach). The slides work on different diameters with the same feed rate.
- B<0: Slide with larger number leads
- B>0: Slide with smaller number leads

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

Program at least NS or NS, NE and P.



- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in Z direction, then in X).
3. Moves at feed rate to target point Z.
4. Depending on H:
   - H=0: Machines the contour outline
   - H=1 or 2: Retracts at 45°
5. Returns at rapid traverse and approach for next pass.
6. Repeats 3 to 5 until target point X has been reached.
7. If required, repeats 2 to 6 until all areas have been machined.
8. If H=1: Smooths the contour
9. Retracts as programmed in Q.

Use as 4 axis cycle

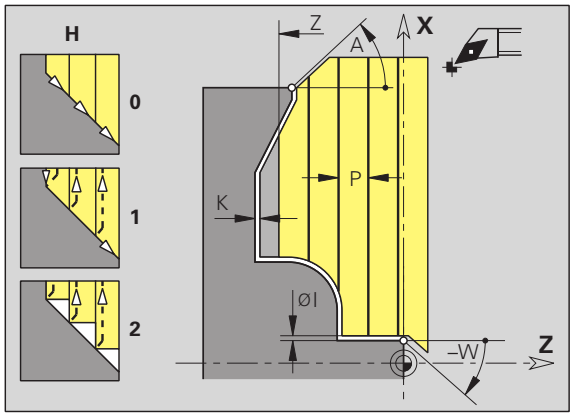
- **Same diameter:** Both slides start simultaneously.
- **Differing diameters:**
  - The second slide starts when the leading slide has reached lead B. This is synchronized at every step.
  - Each slide advances by the calculated depth of cut.
  - If the slides do not have to execute the same number of cuts, the leading slide executes the last cut.
  - With constant surface speed, the cutting speed depends on the speed of the leading slide.
  - The leading tool does not retract until the subsequent tool is ready for use.



On 4 axis cycles, ensure that the tools are identical (tool type, cutting edge radius, cutting edge angle, etc.).

Face roughing G820

G820 machines the contour area from NS to NE defined by NS, NE. If required, the area to be machined is divided into several sections (example: with contour valleys).



Parameters

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P Maximum infeed

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

E Plunging behavior

- E=0: Descending contours are not machined.
- E>0: Plunging feed rate
- No input: Feed rate reduction depending on the plunge angle – maximum 50%

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

H Type of departure (default: 0)

- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45° – no contour smoothing

A Approaching angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

Parameters

W Departure angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

Q Type of retraction at cycle end (default: 0)

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

D Omit elements. The following undercuts, relief turns and recesses are not run (default: 0):

	G22	G23 H0	G23 H1	G25 H4	G25 H5/6	G25 H7 to H9
D=0	•	•	•	•	•	•
D=1	•	•	•	–	–	–
D=2	•	•	–	•	•	•
D=3	•	•	–	–	–	–
D=4	•	•	–	•	•	–

"•": Do not machine the elements

B Slide lead with 4-axis machining

- B=0: Both slides work at the same diameter—at double feed rate
- B<>0: Distance to the leading slide (the approach). The slides work on different diameters with the same feed rate.
- B<0: Slide with larger number leads
- B>0: Slide with smaller number leads

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

Program at least NS or NS, NE and P.

- The **tool radius compensation:** is active.
- A **G57 oversize** Enlarges the contour (also inside contours)
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in X direction, then in Z).
3. Moves at feed rate to target point X.
4. Depending on H:
   - H=0: Machines the contour outline
   - H=1 or 2: Retracts at 45°
5. Returns at rapid traverse and approaches for next pass.
6. Repeats 3 to 5 until target point Z has been reached.
7. If required, repeats 2 to 6 until all areas have been machined.
8. If H=1: Smooths the contour
9. Retracts as programmed in Q.

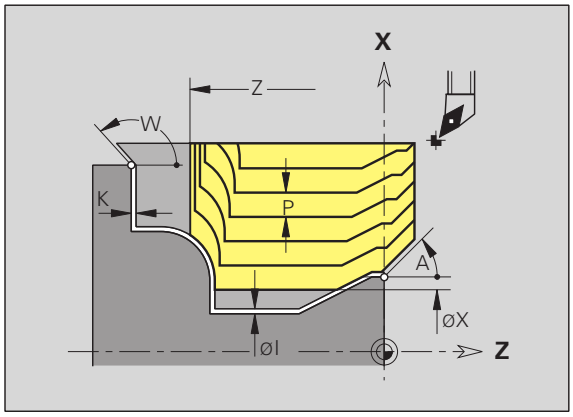
Use as 4 axis cycle

- **Same diameter:** Both slides start simultaneously.
- **Differing diameters:**
  - The second slide starts when the leading slide has reached lead B. This is synchronized at every step.
  - Each slide advances by the calculated depth of cut.
  - If the slides do not have to execute the same number of cuts, the leading slide executes the last cut.
  - With constant surface speed, the cutting speed depends on the speed of the leading slide.
  - The leading tool does not retract until the subsequent tool is ready for use.

On 4 axis cycles, ensure that the tools are identical (tool type, cutting edge radius, cutting edge angle, etc.).

Contour-parallel roughing G830

G830 machines the area parallel to the contour from NS to NE as defined by NS, NE. If required, the area to be machined is divided into several sections (example: with contour valleys).



Parameters

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P Maximum infeed

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

A Approach angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

W Departing angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

Q Type of retraction at cycle end (default: 0)

- Q=0: Return to starting point (first X, then Z direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

Parameters

D Omit elements. The following undercuts, relief turns and recesses are not run (default: 0):

	G22	G23 H0	G23 H1	G25 H4	G25 H5/6	G25 H7 to H9
D=0	•	•	•	•	•	•
D=1	•	•	•	–	–	–
D=2	•	•	–	•	•	•
D=3	•	•	–	–	–	–
D=4	•	•	–	•	•	–

"•": Do not machine the elements

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

Program at least NS or NS, NE and P.

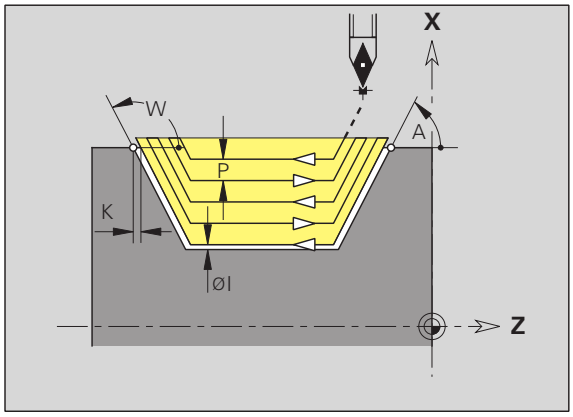
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
3. Executes the first cut (roughing).
4. Returns at rapid traverse and approaches for next pass.
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. Retracts as programmed in Q.

Contour-parallel with neutral tool G835

G835 machines the contour area defined by NS, NE parallel to the contour and bidirectionally. If required, the area to be machined is divided into several sections (example: with contour valleys).



Parameters

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

P Maximum infeed

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

A Approach angle (reference: Z-axis) – (default: 0°/180°, parallel to Z-axis)

W Departing angle (reference: Z-axis) – (default: 90°/270°; perpendicular to Z-axis)

Q Type of retraction at cycle end (default: 0)

- Q=0: Return to starting point (first X, then Z direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding arc is machined—not the basic element (prerequisite: Contour section with one element)

Parameters

D Omit elements. The following undercuts, relief turns and recesses are not run (default: 0):

	G22	G23 H0	G23 H1	G25 H4	G25 H5/6	G25 H7 to H9
D=0	•	•	•	•	•	•
D=1	•	•	•	–	–	–
D=2	•	•	–	•	•	•
D=3	•	•	–	–	–	–
D=4	•	•	–	•	•	–

"•": Do not machine the elements

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

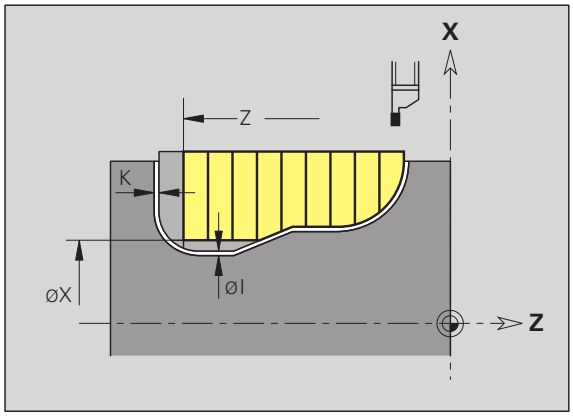
Program at least NS or NS, NE and P.

- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
3. Executes the first cut (roughing).
4. Approaches for the next pass and execute the next cut (roughing) in the opposite direction.
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. Retracts as programmed in Q.

Recessing G860



G860 machines the contouring area axially/radially from NS to NE as defined by NS, NE. The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections (example: with contour valleys).

Parameters

NS Start block number

- Beginning of the contour section, or
- Reference to a G22/G23 Geo recess

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.
- NE is inapplicable if the contour is defined by G22/G23-Geo

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

Q Action (default: 0)

- Q=0: Roughing and finishing
- Q=1: Only roughing
- Q=2: Only finishing

X Cutting limit in X direction (diameter value) – (default: no cutting limit)

Z Cutting limit in Z direction (default: no cutting limit)

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining

E Finishing feed rate (default: active feed rate)

H Type of retraction at cycle end (default: 0)

- H=0: Return to starting point
  - Axial recess: First Z, then X direction
  - Radial recess: First X, then Z direction
- H=1: Positions in front of the finished contour
- H=2: Retracts to safety clearance and stops

The CNC PILOT uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Program at least NS or NS, NE.

Number of cutting passes:

Maximum offset = SBF * cutting width

(SBF: See Machining Parameter 6)



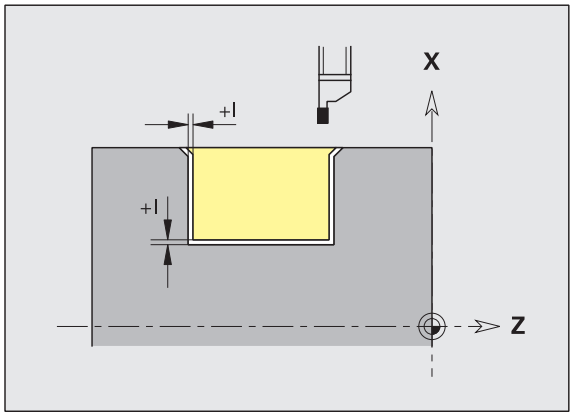
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run (where Q=0 or 1)

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes first cut (roughing).
4. Returns at rapid traverse and approaches for next pass.
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. If Q=0: Finish-machines the contour.

Recessing cycle G866

G866 generates a recess defined by G22-Geo. The CNC PILOT uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.



Parameters

NS Block number (reference to G22-Geo)

I Oversize for roughing (default: 0)

- I=0: Recess is made in one work step
- I>0: The first operation is roughing, the second finishing

E Dwell time (default: Time for one spindle revolution)

- If I=0: For every recess
- If I>0: Only for finishing

Number of cutting passes:

Maximum offset = SBF * cutting width

(SBF: See Machining Parameter 6)



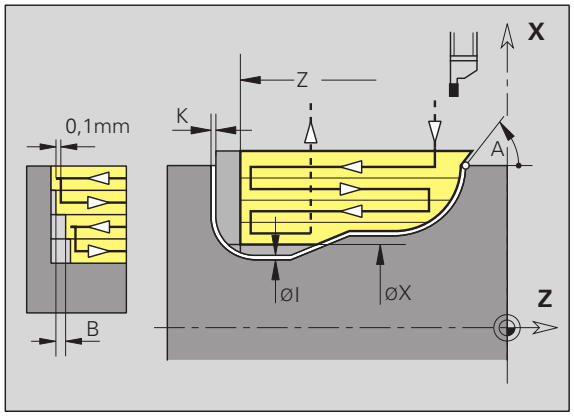
- The **tool radius compensation:** is active.
- An **Oversize** is not taken into account.

Cycle run

1. Calculates the number of cutting passes.
2. Approaches workpiece from starting point for first pass.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes the first cut according to I.
4. Returns at rapid traverse and approaches for next pass.
5. If I=0: Dwells for time E
6. Repeats 3 to 4 until the complete recess has been machined.
7. If I>0: Finish machines the contour

Recess turning cycle G869

G869 machines the contouring area axially/radially from NS to NE as defined by NS, NE. The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements. The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.



Parameters

NS Start block number

- Beginning of the contour section, or
- Reference to a G22/G23-Geo recess

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.
- NE is inapplicable if the contour is defined by G22/G23-Geo

P Maximum infeed

R Turning depth compensation for finishing (default: 0)

I Oversize in X direction (diameter value) – (default: 0)

K Oversize in Z direction (default: 0)

X Cutting limit (diameter value)—(default: no cutting limit)

Z Cutting limit (default: no cutting limit)

A Approach angle (default: opposite to recessing direction)

W Departure angle (default: opposite to recessing direction)

Q Action (default: 0)

- Q=0: Roughing and finishing
- Q=1: Only roughing
- Q=2: Only finishing

U Unidirectional turning (default: 0)

- U=0: The roughing passes are bidirectional.
- U=1: The roughing passes are unidirectional (from NS to NE)

H Type of retraction at cycle end (default: 0)

- H=0: Return to starting point (axial recess: first direction Z, then X; radial recess: first X direction, then Z)
- H=1: Positions in front of the finished contour
- H=2: Retracts to safety clearance and stops

Parameters

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining

O Recessing feed rate (default: active feed rate)

E Finishing feed rate (default: active feed rate)

B Offset width (default: 0)

The CNC PILOT uses the tool definition to distinguish between radial and axial recesses.

Program at least NS or NS, NE and P.

Turning depth compensation R: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with turning depth compensation factor. The value is usually determined empirically.

Offset width B: After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the offset width B. Each time the system switches on this side, the path is reduced by B—in addition to the previous offset. The total offset is limited to 80% of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the CNC PILOT reduces the programmed offset width. After clearance roughing, the remaining material is removed with a single cut.



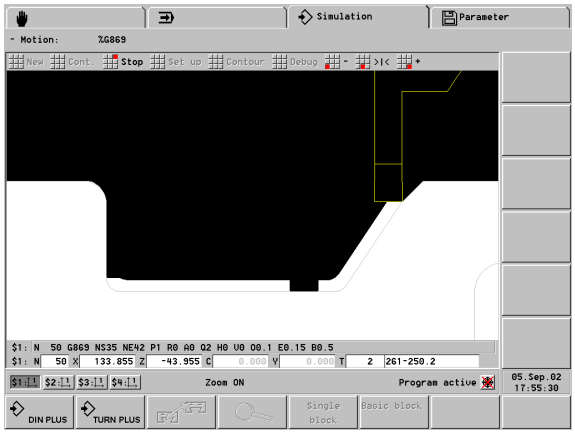
- G869 requires **tools** of the type 26*.
- The **tool radius compensation:** is active.
- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

Cycle run (where Q=0 or 1)

1. Calculates the areas to be machined and the cutting segmentation.
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account.
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes the first cut (recessing).
4. Machines perpendicularly to recessing direction (turning).
5. Repeats 3 to 4 until the complete area has been machined.
6. If required, repeats 2 to 5 until all areas have been machined.
7. If Q=0: Finish-machines the contour.

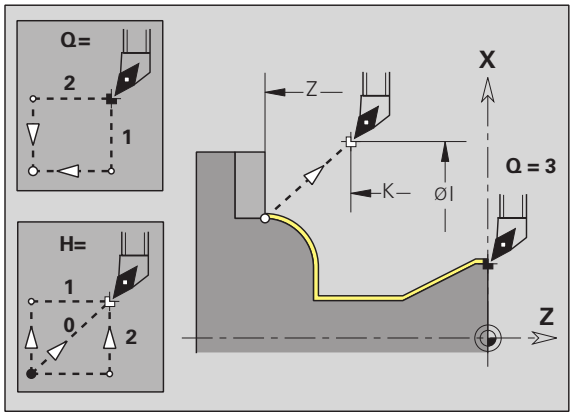
Machining information:

- **Transition from turning to recessing:** Before the transition from turning to recessing, the CNC PILOT retracts the tool by 0.1 mm. Thus an offset cutting edge is adjusted for the recessing operation, independent of “offset width B.”
- **Inside radii and chamfers:** Depending on the recessing width and the radii of rounding arcs, single cuts preventing a "fluid transition" from recessing to turning are executed before the rounding is machined. This prevents damage to the tool.
- **Edges:** Edges are recessed. This prevents residual rings.



Finish contour G890

G890 finishes the contour area defined by NS, NE in one pass and takes chamfers/rounding arc into account. The operation proceeds from NS to NE.



Parameters

NS Starting block number (beginning of contour section)

NE End block number (end of contour section)

- NE not programmed: The contour element NS is machined in the direction of contour definition.
- NS=NE programmed: The contour element NS is machined opposite to the direction of contour definition.

E Plunging behavior

- E=0: Descending contours are not machined.
- E>0: Plunging feed rate
- No input: Machine falling contours at programmed feed rate

V Identifier beginning/end (default: 0)

A chamfer/rounding arc is machined:

- V=0: At start and end
- V=1: At start
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding arc is machined, not the basic element (prerequisite: Contour section with one element)

Q Type of approach (default: 0)

- Q=0: Automatic selection—the CNC PILOT checks:
  - Diagonal approach
  - First X, then Z direction
  - Equidistant around the barrier
  - Omission of the first contour element if the starting position is inaccessible
- Q=1: First X, then Z direction
- Q=2: First Z, then X direction
- Q=3: No approach—tool is located near the starting point of the contour area.
- Q=4: Residual finishing

Omit-codes for recesses and undercuts		
G call	**Function**	**D code**
G22	Recess for sealing ring	512
G22	Recess for guard ring	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G23 H4	Undercut type U	32.768
G23 H5	Undercut type E	65.536
G23 H6	Undercut type F	131.072
G23 H7	Undercut type G	262.744
G23 H8	Undercut type H	524.288
G23 H9	Undercut type K	1.048.576
Add the codes if you want to skip several elements.		

Parameters

H Type of retraction (default: 3)

Tool backs off at 45° against the machining direction and moves as follows to the position I, K:

- H=0: Diagonal
- H=1: First X, then Z direction
- H=2: First Z, then X direction
- H=3: Remains at safety clearance
- H=4: No traverse—tool remains on the end coordinate

X Cutting limit (diameter value)—(default: no cutting limit)

Z Cutting limit (default: no cutting limit)

D Omit elements (default: 1). Use the omit codes listed in the table at right to omit individual elements, or the following codes to skip execution of recesses, undercuts and relief turns.

	G22	G23 H0	G23 H1	G25 H4	G25 H5/6	G25 H7/8	G25 H9
D=0	•	•	•	•	•	•	•
D=1	•	•	–	–	–	–	–
D=2	•	•	–	•	•	•	•
D=3	•	•	•	–	–	–	–
D=4	•	•	–	–	•	–	–
D=5	•	•	–	–	–	–	•
D=6	•	•	–	•	–	•	•
D=7	–	–	–	–	–	–	–

"•": Do not machine the elements

I End point that is approached at the end of the cycle (diameter value)

K End point that is appropriate at the end of the cycle

O Feed rate reduction for circular elements (default: 0)

- O=0: Feed rate reduction active
- O=1: No feed rate reduction

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

Undercuts are machined if they are programmed and if tool geometry permits.

Automatic feed rate reduction for chamfers/rounding arcs:

- Peak-to-valley height or feed rate with G95-Geo are programmed: No automatic feed reduction.
- Surface roughness or feed rate are not programmed: Automatic feed rate reduction; the chamfer/rounding arc is machined with at least 3 revolutions
- For chamfers/rounding arcs which, as a result of their size, are machined with at least three revolutions, the feed rate is not reduced automatically.

Feed rate reduction for circular elements: The tool radius compensation (TRC) results under certain conditions to a feed rate reduction for circular elements (see "Tool-Tip and Cutter Radius Compensation" on page 196). You can switch this feed rate reduction off with O.

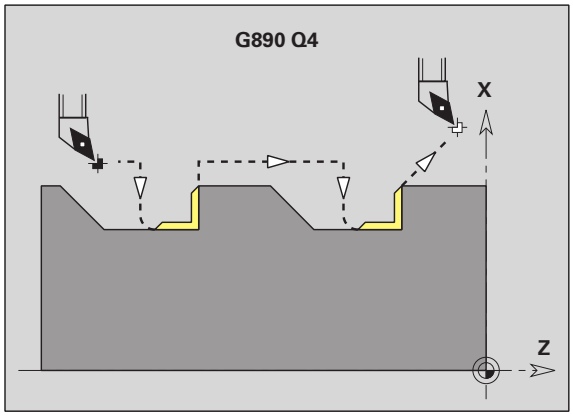


- A **G57 oversize** enlarges the contour (also inside contours).
- A **G58 oversize**
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57/G58 oversizes** are deleted after cycle end.

You activate the **residual finishing** with Q=4 (example: hollowing with finishing tools that machine in the direction opposite to that defined). The CNC PILOT knows the areas that have already been machined and does not machine them again. If Q=4, you cannot influence the approach type. It is determined by the finishing cycle.



During **residual finishing** (G890 – Q4), the CNC PILOT checks whether the tool can move into the contour valley without a collision. The collision check is based on tool parameter "width dn."



4.21 Simple Turning Cycles

End of cycle G80

G80 concludes a fixed cycle.

Simple longitudinal roughing G81

G81 roughs the contour area defined by the current tool position and X, Z. If you wish to machine an oblique cut, you can define the angle with I and K.

Parameters

X Target point contour (diameter)

Z Target point contour

I Maximum infeed in X direction

- I<0: With machining contour outline
- I>0: Without machining contour outline

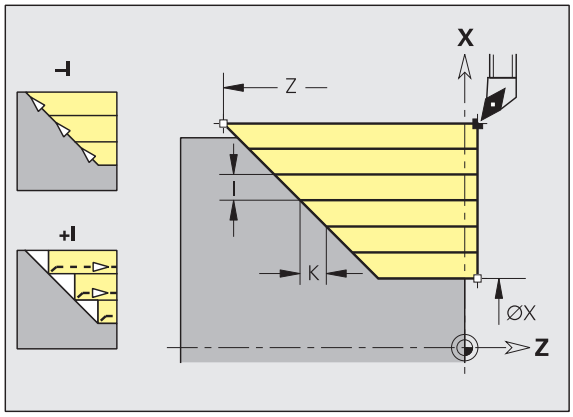
K Offset in Z direction (default: 0)

Q G function for infeed (default: 0)

- 0: Infeed with G0 (rapid traverse)
- 1: Infeed with G1 (feed rate)

The CNC PILOT uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated so that an abrasive cut is avoided and the calculated infeed distance is <= I.

- **Programming X, Z:** Absolute, incremental or modal
- The **tool radius compensation** is not active.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize** is not taken into account.



Example: G81

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G81 X100 Z-70 I4 K4 Q0
N4 G0 X100 Z2
N5 G81 X80 Z-60 I-4 K2 Q1
N6 G0 X80 Z2
N7 G81 X50 Z-45 I4 Q1
. . .
```

Cycle run

1. Calculates the number of cutting passes.
2. Approaches workpiece for first pass from starting point on paraxial path.
3. Moves at feed rate to target point Z.
4. Depending on algebraic sign of I:
   - <0: Machines contour outline
   - I>0: Retracts by 1 mm at 45°
5. Returns at rapid traverse and approaches for next pass.
6. Repeats 3 to 5 until target point X has been reached.
7. Move to:
   - X: Last retraction coordinate
   - Z: Starting point of cycle

Simple face roughing G82

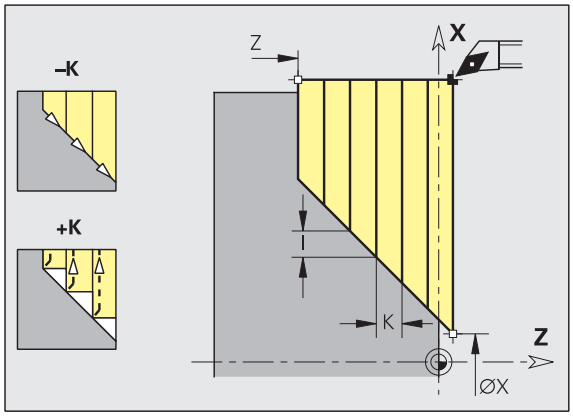
G82 roughs the contour area defined by the current tool position and X, Z. If you wish to machine an oblique cut, you can define the angle with I and K.

Parameters

X	Target point contour (diameter)
Z	Target point contour
I	Offset in X direction (default: 0)
K	Maximum infeed
	■ K<0: With machining contour outline
	■ K>0: Without machining contour outline
Q	G function for infeed (default: 0)
	■ 0: Infeed with G0 (rapid traverse)
	■ 1: Infeed with G1 (feed rate)

The CNC PILOT uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated so that an abrasive cut is avoided and the calculated infeed distance is <= K.

- **Programming X, Z:** Absolute, incremental or modal
- The **tool radius compensation** is not active.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize** is not taken into account.



Example: G82

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G82 X20 Z-15 I4 K4 Q0
N4 G0 X120 Z-15
N5 G82 X50 Z-26 I2 K-4 Q1
N6 G0 X120 Z-26
N7 G82 X80 Z-45 K4 Q1
. . .
```

Cycle run

1. Calculates the number of cutting passes (infeeds).
2. Approaches workpiece for first pass from starting point on paraxial path.
3. Moves at feed rate to target point X.
4. Depending on algebraic sign of K:
   - K<0: Machines contour outline
   - K>0: Retracts by 1 mm at 45°
5. Returns at rapid traverse and approaches for next pass.
6. Repeats 3 to 5 until target point Z has been reached.
7. Move to:
   - X: Starting point of cycle
   - Z: Last retraction coordinate

Simple contour repeat cycle G83

G83 carries out the functions programmed in the following blocks (simple traverses or cycles without a contour definition) more than once. G80 ends the machining cycle.

Parameters

X	Contour target point (diameter)—(default: Load the last X coordinate)
Z	Contour target point—(default: Load the last Z coordinate)
I	Maximum infeed in X direction (radius)—(default: 0)
K	Maximum infeed in Z direction (default: 0)

If the number of infeeds differs for the X and Z axes, the tool first advances in both axes with the programmed values. The infeed is set to zero if the target value for one direction is reached.

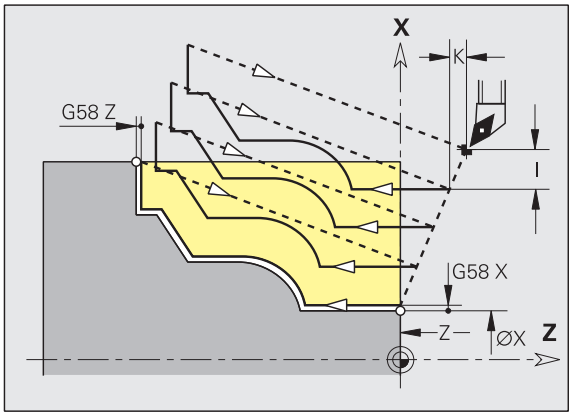
Programming:

- G83 is alone in the block
- G83 must not be programmed with K variables
- G83 must not be nested, not even by calling subprograms

- The **tool radius compensation** is not active. You can program the TRC separately with G40 to G42.
- **Safety clearance** after each pass: 1 mm
- A **G57 oversize**
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58 oversize**
  - Is taken into account if you work with TRC
  - Remains effective after cycle end

Cycle run

1. Starts the cycle execution from the current tool position.
2. Advances by the infeed distance defined in I and K.
3. Executes the machining operation which is programmed in the blocks after G83, taking the distance from the tool position to the contour start point as an "oversize."
4. Returns on a diagonal path.
5. Repeats 2 to 4 until the contour target point has been reached.
6. Returns to the starting point of the cycle.



Example: G83

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G83 X80 Z0 I4 K0.3
N4 G0 X80 Z0
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7 G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X110
N13 G0 Z2
N14 G80
```

Danger of collision!

After each pass, the tool returns on a diagonal path before it advances for the next pass. If required, program an additional rapid traverse path to avoid a collision.

Undercut cycle G85

With the function G85, you can machine undercuts according to DIN 509 E, DIN 509 F and DIN 76 (thread undercut). The CNC PILOT determines the type of undercut using K.

Parameters

X Target point (diameter)

Z Target point

I Depth (radius)

- DIN 509 E, F: Finishing oversize (default: 0)
- DIN 76: Undercut depth

K Undercut width and **type of undercut**

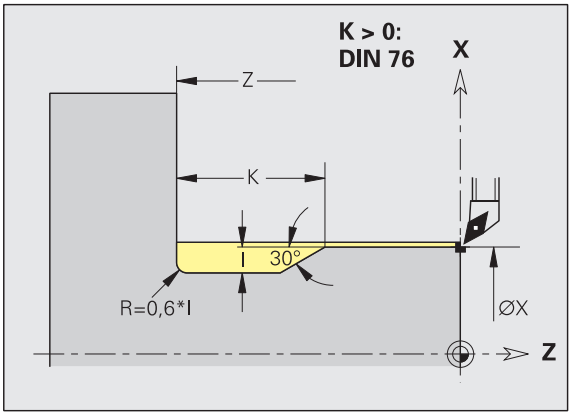
- K No input: DIN 509 E
- K=0: DIN 509 F
- K>0: Undercut width for DIN 76

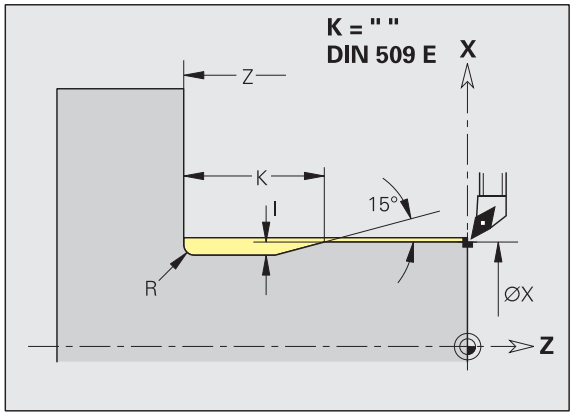
E Reduced feed for machining the undercut (default: active feed rate)

Also see the following tables

G85 machines the adjoining cylinder if you position the tool to diameter X "in front of" the cylinder.

The undercut rounding arcs are executed with the radius 0.6 * I.



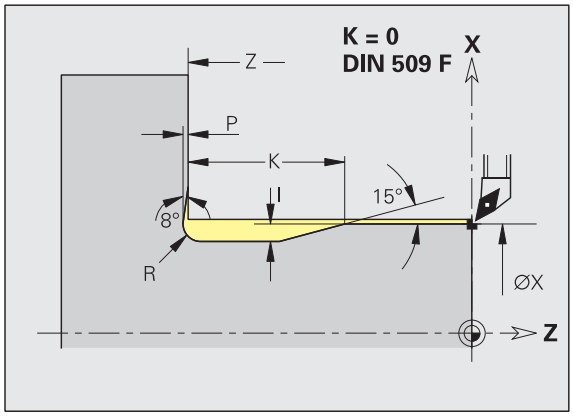


Parameters for undercut DIN 509 E			
Diameter	**I**	**K**	**R**
<= 18	0,25	2	0,6
> 18 – 80	0,35	2,5	0,6
> 80	0,45	4	1

Parameters for undercut DIN 509 E				
Diameter	**I**	**K**	**R**	**P**
<= 18	0,25	2	0,6	0,1
> 18 – 80	0,35	2,5	0,6	0,2
> 80	0,45	4	1	0,3

- I = undercut depth
- K = undercut width
- R = undercut radius
- P = face depth
- **Undercut angle** for undercuts according to DIN 509 E and F: 15°
- **Transverse angle** for an undercut according to DIN 509 F: 8°

- The **tool radius compensation** is not active.
- **Oversizes** are not taken into account.



Example: G85

```
. . .
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X62 Z2
N3 G85 X60 Z-30 I0.3
N4 G1 X80
N5 G85 X80 Z-40 K0
N6 G1 X100
N7 G85 X100 Z-60 I1.2 K6 E0.11
N8 G1 X110
. . .
```

Recessing G86

G86 machines simple radial and axial recesses with chamfers. From the tool position, the CNC PILOT determines whether a radial or axial recess, or an inside or outside recess is to be machined.

Parameters

X Floor corner point (diameter)

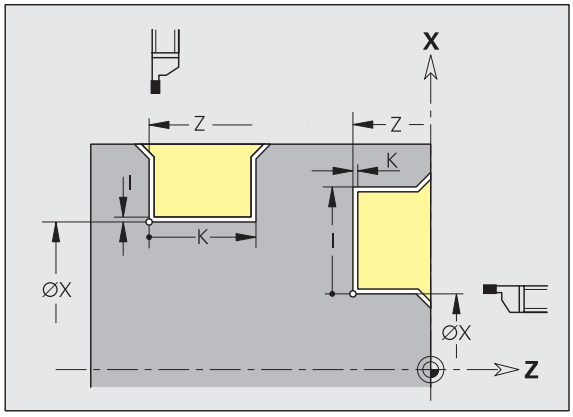
Z Floor corner point

I Radial recess: Oversize

- I>0: Oversize (roughing and finishing)
- I=0: No finishing

Axial recess: Recess width

- I>0: Recess width
- No input: Recess width = tool width



Parameters

K Radial recess: Recess width

- K>0: Recess width
- No input: Recess width = tool width

Axial recess: Oversize

- K>0: Oversize (roughing and finishing)
- K=0: No finishing

E Dwell time (for chip breaking)—(default: length of time for one revolution)

- With finishing oversize: Only for finishing
- Without finishing oversize: For every recess

"Oversize" programmed: First roughing, then finishing

G86 machines chamfers at the sides of the recess. If you do not wish to cut the chamfers, you must position the tool at a sufficient distance from the workpiece. Calculate the starting position XS (diameter) as follows:

XS = XK + 2 * (1.3 – b)

XK: Contour diameter

b: Chamfer width



- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.

Example: G86

```
. . .
N1 T3 G95 F0.15 G96 S200 M3
N2 G0 X62 Z2
N3 G86 X54 Z-30 I0.2 K7 E2          [radial]
N4 G14 Q0
N5 T8 G95 F0.15 G96 S200 M3
N6 G0 X120 Z1
N7 G86 X102 Z-4 I7 K0.2 E1          [axial]
. . .
```

Cycle run

1. Calculates the number of cutting passes.
   Maximum offset SBF * cutting width
   (SBF: See Machining Parameter 6)
2. Approaches to clearance height at rapid traverse on paraxial path.
3. Executes the first cut, taking finishing oversize into account.
4. Without finishing oversize: Dwells for the time period E
5. Retracts and approaches for next pass.
6. Repeats 2 to 4 until the complete recess has been machined.
7. With finishing oversize: Finish-machines the recess
8. Returns paraxially to starting point at rapid traverse.

Radius cycle G87

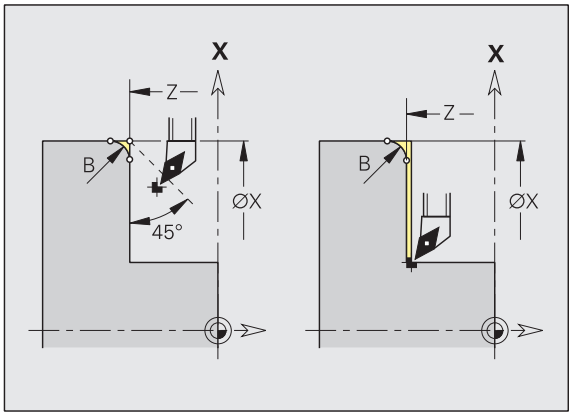
G87 machines transition radii at orthogonal, paraxial inside and outside corners. The direction is taken from the position/machining direction of the tool.

Parameters

X	Corner point (diameter)
Z	Corner point
B	Radius
E	Reduced feed rate (default: active feed)

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.



Example: G87

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G87 X84 Z0 B2                    [radius]
```

Chamfer cycle G88

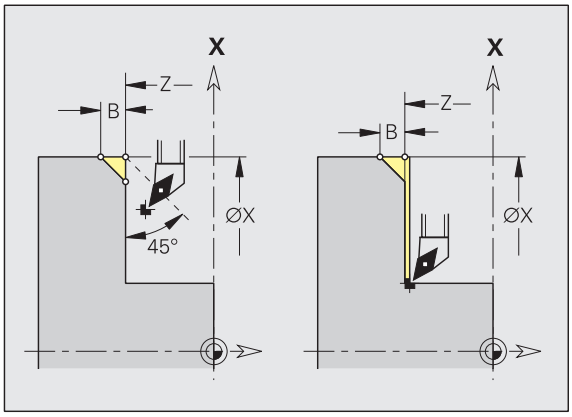
G88 machines chamfers at orthogonal, paraxial outside corners. The direction is taken from the position/machining direction of the tool.

Parameters

X	Corner point (diameter)
Z	Corner point
B	Chamfer width
E	Reduced feed rate (default: active feed)

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

- The **tool radius compensation:** is active.
- **Oversizes** are not taken into account.



Example: G88

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G88 X84 Z0 B2                    [chamfer]
```

4.22 Thread Cycles

Overview of threading cycles

- G31 machines simple threads, successions of threads and multi-start threads with G24, G34 or G37 Geo (see "Thread cycle G31" on page 240). G31 does not switch the velocity feedforward. If you want to work without velocity feedforward you can switch it off before the thread cycle.
- G32 cuts a simple thread in any desired direction and position (see "Simple thread cycle G32" on page 242). G32 is used to deactivate velocity feedforward.
- G33 conducts a single thread cut. The direction of the single thread cut is as desired (see "Thread single path G33" on page 244). G33 does not switch the velocity feedforward. If you want to work without velocity feedforward you can switch it off before the thread cycle.

Smooth threading: With smooth threading, the CNC PILOT accelerates over cubic acceleration ramps. The smooth threading prevents vibration during thread cutting on lathes with direct drives (see "Thread switch G933" on page 239).

Thread switch G933

With **smooth threading** the CNC PILOT accelerates during thread run-in, thread run-out and during direction reversal (successive threads) over cubic acceleration ramps. The smooth threading prevents vibration during thread cutting on lathes with direct drives.

Parameters

Q Thread switch

- Q=0: Smooth threading off
- Q=1: Smooth threading on

G933 switches the smooth threading on/off. G933 is a modal function. It can be programmed at any location, even in the G33 block. With program start, with M30 and M99 the smooth threading is switched off.

The smooth threading is supported by NC software 368 650-22 and later. As of software version 368 650-23, the smooth threading can be activated on a sustained basis by parameter. Set bit 5 of the expansion stage code (MP 1103, ..).

Thread cycle G31

G31 machines simple threads, successions of threads and multi-start threads with G24-, G34- or G37-Geo. The CNC PILOT uses the tool definition to distinguish between external and internal threads.

Parameters

NS Block number (reference to basic element G1 Geo for successions of threads: block number of the first basic element)

I Maximum infeed

B Run-in length—no input: Run-in length is calculated from adjacent undercuts or recesses. If they does not exist, the thread starting length from machining parameter 7 applies.

P Run-out length—no input: Run-out length is calculated from adjacent undercuts or recesses. If they does not exist, the thread run-out length from machining parameter 7 applies.

D Cutting direction (reference: definition direction of basic element)—(default: 0)

- D=0: Same direction
- D=1: Opposite direction

V Type of infeed (default: 0)

- V=0: Constant cross section for all cuts
- V=1: Constant infeed
- V=2: With distribution of remaining cuts First infeed = Remainder of the division of thread depth/cutting depth. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
- V=3: Infeed is calculated from the pitch and spindle speed

H Type of offset for smoothing the thread flanks (default: 0)

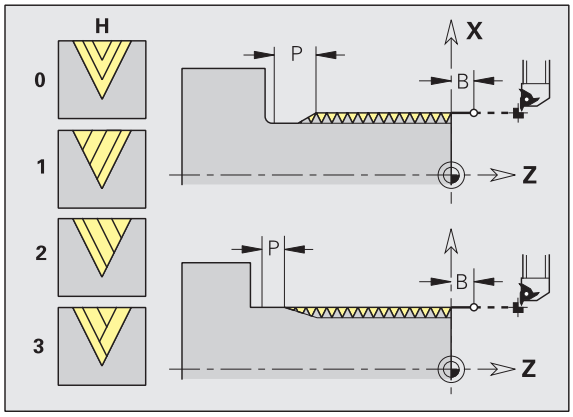
- H=0: Without offset
- H=1: Offset from left
- H=2: Offset from right
- H=3: Tool is offset alternately from the right and left

Q Number of air cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)

C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)

Run-in length B: The slide requires a run-in distance at the start of thread in order to accelerate to the programmed feed rate before starting the actual thread.

Run-out length P: The slide needs an overtravel at the end of the thread to decelerate again. Remember that the paraxial line P needs overtravel even with an oblique thread run-out.



Example: G31, part 1

```
. . .
FERTIGTEIL [FINISHED PART]
N   2 G0 X16 Z0
N   3 G52 P2 H1
N   4 G95 F0.8
N   5 G1 Z-18
N   6 G25 H7 I1.15 K5.2 R0.8 W30
N   7 G37 Q12 F2 P0.8 A30 W30
. . .
BEARBEITUNG [MACHINING]
N  33   G14 Q0 M108
N  30   T9 G97 S1000 M3
N  34   G47 P2
N  35   G31 NS5 B5 P0 V0 H1
N  36   G0 X110 Z20
N  38   G47 M109
. . .
```

You can calculate the minimum run-in and run-out length with the following equation.

Smooth threading switched off
Run-in length: B = 0.75 * (F*S)² / a + 0.15
Run-out length: P = 0.75 * (F*S)² / e + 0.15
Smooth threading switched on
Run-in length: B = 0.75 * (F*S)² / a * 0.66 + 0.15
Run-out length: P = 0.75 * (F*S)² / e * 0.66 + 0.15

- F: Thread pitch in mm/revolution
- S: Speed in **revolutions/second**
- a, e: Acceleration in mm/s² (see "Acceleration at block start/block end" in MP 1105, ...)

Starting angle C: At the end of the "run-in path B" the spindle is at the "starting angle C" position. Therefore, if the thread is to start exactly at the starting angle, position the tool by the run-in length or a multiple of it in front of the beginning of the thread.

Look-ahead: G31 **does not** switch the look-ahead off. You can switch the look-ahead off and on again in separate NC blocks (see "Velocity feedforward G918" on page 307).

The individual thread cuts are calculated from the thread depth, maximum approach I and type of approach V.

You influence the thread cutting with the smooth threading (see "Thread switch G933" on page 239).



- Feed rate stop becomes effective only at the end of a thread cut.
- Feed rate override is not effective.
- Do not use the spindle override if the feedforward is switched off!



Danger of collision!

- An excessive run-out length P might cause a collision. You can check the run-out length during the simulation.
- The spindle reference is derived from the feed per revolution that was programmed last.

Cycle run

1. Calculates the number of cutting passes.
2. Returns diagonally to the internal starting point at rapid traverse. This point lies in front of the "starting point of thread" by the run-in length B. With H=1 (or 2, 3) the current offset is taken into account for calculating the internal starting point.

   The internal starting point is calculated on the basis of the tool tip.
3. Accelerates to feed rate (line B).
4. Executes a thread cut.
5. Decelerates (line P).
6. Retracts to safety clearance, returns at rapid traverse, and approaches for next pass. For multiple threads, the same rate of cut is used for each thread turn, before the next infeed motion is executed.
7. Repeats 3 to 6 until the complete thread has been cut.
8. Executes air cuts.
9. Returns to internal starting point.

Simple thread cycle G32

G32 cuts a simple thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread).

Parameters

X End point of thread (diameter)

Z End point of thread

F Thread pitch

P Thread depth

I Maximum cutting depth

B Remainder cuts (default: 0)

- B=0: The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
- B=1: Without distribution of remaining cut

Q Number of no-load (air) cuts after the last cut (for reducing the cutting pressure in the thread base)—(default: 0)

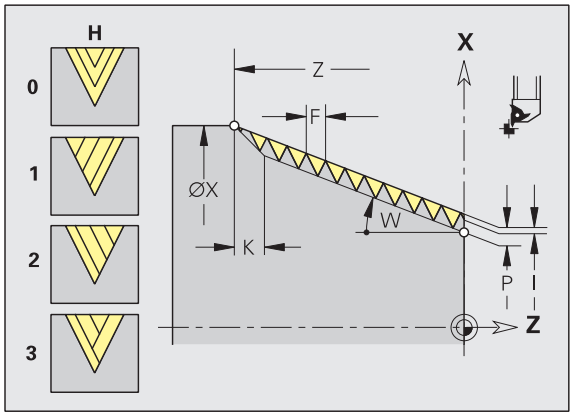
K Run-out length at thread end point (default: 0)

W Taper angle (–45° < W < 45°)—(default: 0)

Position of the taper thread with respect to longitudinal or transverse axis:

- W>0: Rising contour (in machining direction)
- W<0: Falling contour

C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)



Example: G32

```
. . .
N1 T4 G97 S800 M3
N2 G0 X16 Z4
N3 G32 X16 Z-29 F1.5 U-0.9 I0.2    [thread]
. . .
```

Parameters

H Type of offset for smoothing the thread flanks (default: 0)

- H=0: Without offset
- H=1: Offset from left
- H=2: Offset from right
- H=3: Tool is offset alternately from the right and left

The cycle calculates the thread from the thread end point, thread depth and the tool position. The main machining direction of the tool determines whether an internal or an external thread will be machined.

First infeed = Remainder of the division of thread depth/cutting depth.

You influence the thread cutting with the smooth threading (see "Thread switch G933" on page 239).



- Feed rate stop becomes effective only at the end of a thread cut.
- Feed rate and spindle override are not effective.
- Make thread with G95 (feed rate per revolution).
- **Feedforward control** is switched off.

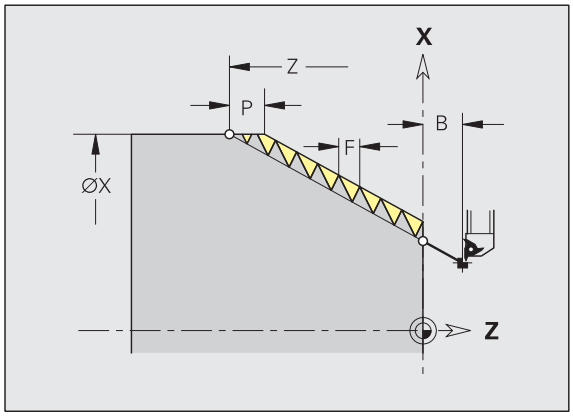
Cycle run

1. Calculates the number of cutting passes.
2. Executes a thread cut.
3. Returns at rapid traverse and approaches for next pass.
4. Repeats 2 to 3 until the complete thread has been cut.
5. Executes air cuts.
6. Returns to starting point.

Thread single path G33

G33 conducts a single thread cut. The direction of the single thread path is as desired (longitudinal, tapered or transverse threads; internal or external threads). You can make successive threads by programming G33 several times in succession.

Position the tool in front of the thread by the run-in length B if the slide must accelerate to the feed rate. And remember the run-out length P **before** the end point of thread if the slide has to be decelerated.



Parameters

X End point of thread (diameter)

Z End point of thread

F Feed per revolution (thread pitch)

B Run-in length (length of the acceleration path)—default :0

P Run-in length (length of the acceleration path)—default: 0

C Starting angle (thread start is defined with respect to rotationally nonsymmetrical contour elements)—(default: 0)

Q Number of the spindle

H Reference direction for thread pitch (default: 0)

- H=0: Feed rate on the Z axis for longitudinal and taper threads up to a max. angle of +45°/–45° to the Z axis
- H=1: Feed rate on the X axis for transverse and taper threads up to a max. angle of +45°/–45° to the X axis
- H=3: Contouring feed rate

E Variable pitch (default: 0)

- E=0: Constant pitch
- E>0: Increases the pitch per revolution by E
- E<0: Decreases the pitch per revolution by E

Run-in length B: The slide requires a run-in distance at the start of thread in order to accelerate to the programmed feed rate before starting the actual thread.

Run-out length P: The slide needs an overtravel at the end of the thread to decelerate again. Remember that the paraxial line P needs overtravel even with an oblique thread run-out

If the thread is made with look-ahead, the CNC PILOT checks the run-in and run-out length. If the paths are shorter than shown in the following equation, the control issues a warning.

You can calculate the minimum run-in and run-out length with the following equation.

Smooth threading switched off
Run-in length: B = 0.75 * (F*S)² / a + 0.15
Run-out length: P = 0.75 * (F*S)² / e + 0.15
Smooth threading switched on

Example: G33

```
. . .
N1 T5 G97 S1100 G95 F0.5 M3
N2 G0 X101.84 Z5
N3 G33 X120 Z-80 F1.5  [thread single path]
N4 G33 X140 Z-122.5 F1.5
N5 G0 X144
. . .
```

Run-in length: B = 0.75 * (F*S)² / a * 0.66 + 0.15

Run-out length: P = 0.75 * (F*S)² / e * 0.66 + 0.15

- F: Thread pitch in mm/revolution
- S: Speed in **revolutions/second**
- a, e: Acceleration in mm/s² (see "Acceleration at block start/block end" in MP 1105, ...)

Starting angle C: At the end of the "run-in path B" the spindle is at the "starting angle C" position.

Feedforward: G31 **does not** switch the feedforward off. You can switch the feedforward off and on again in separate NC blocks (see "Velocity feedforward G918" on page 307).

You influence the thread cutting with the smooth threading (see "Thread switch G933" on page 239).



- Feed rate stop becomes effective only at the end of a thread cut
- Feed rate override is not effective
- Do not use the spindle override if the feedforward is switched off!
- Create thread with G95 (feed rate per revolution)

Cycle run

1. Accelerates to feed rate (line B).
2. Moves to end point of thread – run-out length P.
3. Decelerates (line P) remains at the end point of thread.

4.23 Drilling Cycles

Drilling cycle G71

G71 is used for axial and radial holes using driven or stationary tools for:

- Single hole without contour description
- Hole with contour description (single hole or hole pattern)

Parameters

NS Block number of contour

- Reference to the contour of the hole (G49-, G300- or G310-Geo)
- No input: Single hole without contour description

NF Reference from which the cycle reads the hole positions [1 to 127].

X End point of axial hole (diameter value)

Z End point of radial hole

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

V Feed rate reduction (50 %)—(default: 0)

- V=0 or 2: Reduction at start
- V=1 or 3: Reduction at start and end
- V=4: Reduction at end
- V=5: No reduction

D Retraction speed (default: 0)

- D=0: Rapid traverse
- D=1: Feed rate

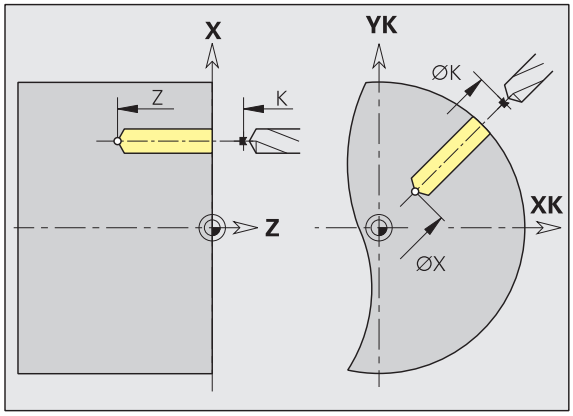
K Retraction plane (radial holes, holes in the YZ plane: diameter)—(default: retract to starting position or to safety clearance)

H1 As of software version 625 952-04:

Spindle brake (H1 is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0

- 0: Activate the spindle brake
- 1: Deactivate the spindle brake

Hole positions that you find with the milling cycles G840 A1 .., G845 A1 .., or G846 A1 .., you can pre-drill with G71 NF.. (see "Milling Cycles" on page 261).



Example: G71

```
. . .
N1 T5 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-25 A5 V2                [drilling]
. . .
```

Feed rate reduction factor:

- Indexable insert drill and twist drill with 180° drilling angle
  - Beginning of hole: No reed rate reduction (even with V=0 or V=1)
  - End of hole: Reduction as of hole end point – 2*safety clearance
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in V
  - End of hole: Reduction as of hole end point – length of first cut – safety clearance
- Length of first cut = tool tip
- Safety clearance: see machining parameter 9 “Drilling” or G47, G147).



- Single hole without contour description: Program X or Z as alternative.
- Hole with contour description: Do not program X, Z.
- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.

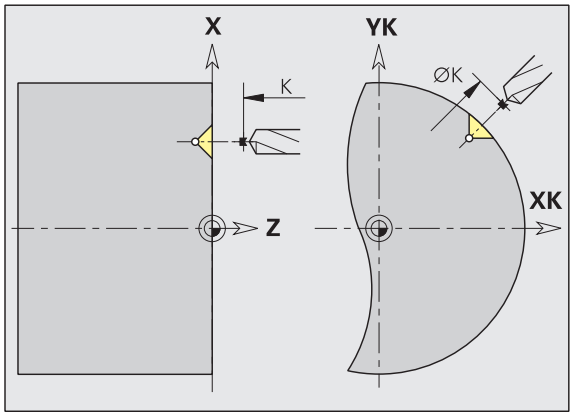
Cycle run

1. **Hole without contour definition:** Tool is located at the starting point (safety distance from the bore hole).
   Hole with contour description: Tool moves at rapid traverse to the starting point:
   - K not programmed: Moves up to the safety clearance
   - K programmed: Moves to the position K and then to the safety clearance
2. Spot drilling. Feed rate reduction depending on V
3. Drilling at feed rate.
4. Through drilling. Feed rate reduction depending on V
5. Retract depending on D at rapid traverse or feed rate.
6. Retraction position:
   - K not programmed: Retraction to the starting point
   - K programmed: Retraction to the position K

Boring, countersinking G72

G72 is used for holes with contour definition (individual hole or hole pattern). Use G72 for the following axial and radial drilling functions using driven or stationary tools:

- Boring
- Countersinking
- Reaming
- NC centering
- Centering



Parameters

NS Block number of contour. Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

D Retraction speed (default: 0)

- D=0: Rapid traverse
- D=1: Feed rate

K Retraction plane (radial holes, holes in the YZ plane: diameter)—(default: to starting position or to safety clearance)

H1 As of software version 625 952-04:

Spindle brake (H1 is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0

- 0: Activate the spindle brake
- 1: Deactivate the spindle brake

Cycle run

1 Moves depending on K at rapid traverse to the starting point:

- K not programmed: Moves up to the safety clearance
- K programmed: Moves to the position K and then to the safety clearance

2 Drills at reduced feed rate (50%).

3 Moves at feed rate to end of hole.

4 Retract depending on D at rapid traverse or feed rate.

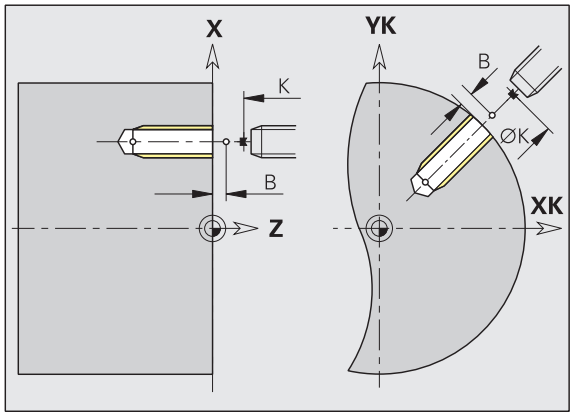
5 Retraction position depends on K:

- K not programmed: Retraction to the starting point
- K programmed: Retraction to the position K

Hole pattern: NS refers to the hole contour, and not the definition of the pattern.

Tapping G73

G73 cuts axial/radial threads using driven or stationary tools. G73 is used for holes with contour definition (individual hole or hole pattern).



Parameters

NS Block number of contour. Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)

B Slope length (default: machining parameter 7 "Thread starting length [GAL]")

S Retraction speed (default: Tapping speed)

K Retraction plane (radial holes, holes in the YZ plane: diameter)—(default: retract to starting position or to safety clearance)

J Retraction length when using floating tap holders (default: 0)

H1 As of software version 625 952-04:

Spindle brake (H1 is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0

- 0: Activate the spindle brake
- 1: Deactivate the spindle brake

The starting position is calculated from the safety clearance and the run-in (slope) length B.

Retraction length J: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retract length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.



- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.
- "Cycle STOP" becomes effective at the end of the tapping operation.
- Feed rate override is not effective.
- Do not use spindle override!

Cycle run

1 Moves at rapid traverse to the starting point:
- K not programmed: Retracts directly to the starting point
- K programmed: Moves to the position K and then to the starting point

2 Moves along run-in length B feed rate (synchronization of spindle and feed drives).

3 Cuts the thread.

4 Retracts with return speed S:
- K not programmed: To the starting point
- K programmed: To the position K

Tapping G36

G36 cuts axial/radial threads using driven or stationary tools. Depending on X/Z, G36 decides whether a radial or axial thread will be machined.

Move to the starting point before G36. G36 returns to the starting position after having cut the thread.

Parameters

X End point of axial hole (diameter value)

Z End point of radial hole

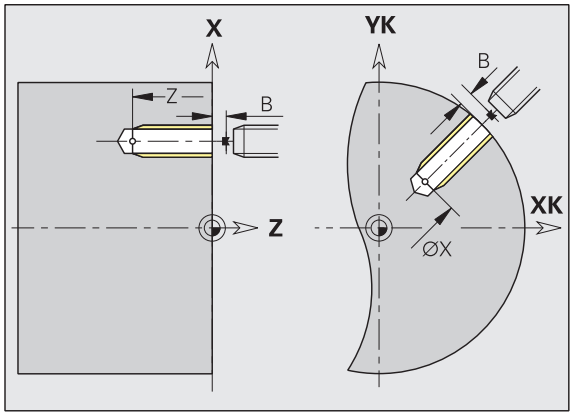
F Feed per revolution: Thread pitch

Q Number of spindle (default: 0—main spindle)

B Run-in length for synchronizing spindle and feed drive

H Reference direction for thread pitch (default: 0)
- H=0: Feed rate on the Z axis
- H=1: Feed rate on the X axis
- H=2: Feed rate on the Y axis
- H=3: Contouring feed rate

S Retraction speed (default: Tapping speed)



Example: G36

```
. . .
N1 T5 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-30
N4 G14 Q0
N5 T6 G97 S600 M3
N6 G0 X0 Z8
N7 G36 Z-25 F1.5 B3 Q0                [tapping]
. . .
```

Type of taps:

- Stationary tap: Main spindle and feed drive are synchronized.
- Driven tap: Driven tool and feed drive are synchronized.



- "Cycle STOP" becomes effective at the end of the tapping operation.
- Feed rate override is not effective.
- Do not use spindle override!
- Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder.

Deep-hole drilling G74

G74 is used for axial and radial holes in several stages using driven or stationary tools.

Parameters

NS Block number of contour

- Reference to the contour of the hole (G49-Geo, G300-Geo or G310-Geo)
- No input: Single hole without contour description

X End point of axial hole (diameter value)

Z End point of radial hole

P 1st hole depth

I Reduction value (default: 0)

B Retraction distance (default: to starting point of hole)

J Minimum hole depth (default: 1/10 of P)

E Period of dwell for chip breaking at end of hole (in seconds)—(default: 0)

V Feed rate reduction (50 %)—(default: 0)

- V=0 or 2: Reduction at start
- V=1 or 3: Reduction at start and end
- V=4: Reduction at end
- V=5: No reduction

D Retraction speed and infeed within the hole (default: 0)

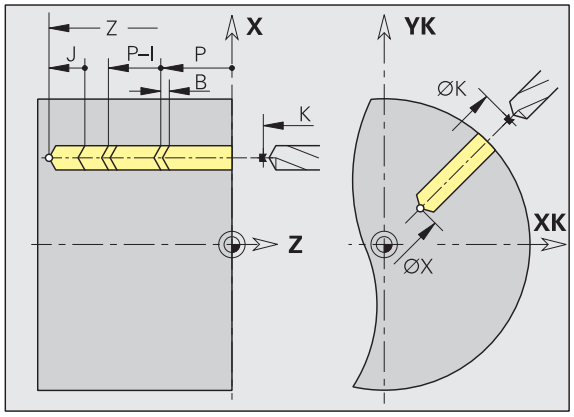
- D=0: Rapid traverse
- D=1: Feed rate

K Retraction plane (radial holes: diameter)—(default: to starting position or to safety clearance)

H1 As of software version 625 952-04:

Spindle brake (H1 is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0

- 0: Activate the spindle brake
- 1: Deactivate the spindle brake



Example: G74

```
. . .
N1 M5
N2 T4 G197 S1000 G195 F0.2 M103
N3 M14
N4 G110 C0
N5 G0 X80 Z2
N6 G74 Z-40 R2 P12 I2 B0 J8        [drilling]
N7 M15
. . .
```

The cycle is used for:

- Single hole without contour description
- Hole with contour description (single hole or hole pattern)

"1st drilling depth P" is used for the first pass. MANUALplus then automatically reduces the drilling depth with each subsequent pass by the reduction value I, however, without falling below the minimum drilling depth J. After each pass, the tool is retracted either by retraction distance B or to the starting point of the hole.

Feed rate reduction:

- Indexable insert drill and twist drill with 180° drilling angle
  - Beginning of hole: No feed rate reduction (even with V=0 or V=1)
  - End of hole: Reduction as of hole end point – 2*safety clearance
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in V
  - End of hole: Reduction as of hole end point—length of first cut—safety clearance
- Length of first cut = tool tip
- Safety clearance: see machining parameter 9 "Drilling" or G47, G147).



- Single hole without contour description: Program X or Z as alternative.
- Hole with contour description: Do not program X, Z.
- Hole pattern: NS refers to the hole contour, and not the definition of the pattern.
- A "feed rate reduction at end" goes into effect only at the last drilling stage

Cycle run

1. **Hole without contour definition:** Tool is located at the starting point (safety distance from the bore hole).
   Hole with contour description: Tool moves at rapid traverse to the starting point:
   - K not programmed: Moves up to the safety clearance
   - K programmed: Moves to the position K and then to the safety clearance
2. Spot drilling. Feed rate reduction depending on V
3. Drill hole in several passes.
4. Through drilling. Feed rate reduction depending on V
5. Retract depending on D at rapid traverse or feed rate.
6. Retraction position depends on K:
   - K not programmed: Retraction to the starting point
   - K programmed: Retraction to the position K

4.24 C-Axis Commands

No. of C axis G119

Use G119 if several C axes are available and the active C axis is switched during machining. With G119 without Q, deselect the old assignment and assign the C-Axis to the slide with G119 Q...

Parameters

Q Number of the C axis (default: 0)

- Q=0: Cancel assignment of C axis to slide
- Q>0: Assign the C axis to the slide

Reference diameter G120

G120 determines the reference diameter of the unrolled lateral surface. Program G120 if you use CY for G110 to G113. G120 is a modal function.

Parameters

X Diameter

Example: G120

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100            [reference diameter]
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
. . .
```

Zero point shift, C axis G152

G152 defines an absolute zero point for the C axis (reference: MP 1005, .. "Reference point, C axis"). The zero point is valid until the end of the program.

Parameters

C Angle (spindle position) of the new C-axis zero point

Example: G152

```
. . .
N1 M5
N2 T7 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30            [zero point of C axis]
N5 G110 C0
N6 G0 X122 Z-50
N7 G71 X100
N8 M15
. . .
```

Standardize C axis G153

G153 resets a traverse angle >360° or <0° to the corresponding angle modulo 360°—without moving the C axis.



G153 is only used for lateral-surface machining. An automatic modulo 360° function is carried out on the face.

4.25 Front/Rear-Face Machining

Rapid traverse on front/rear face G100

G100 moves at rapid traverse along the shortest path to the end point.

Parameters

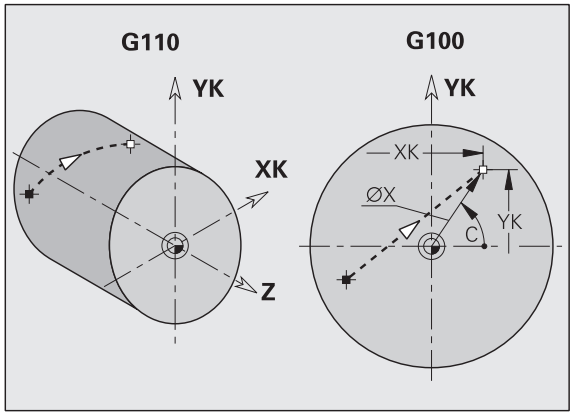
X	End point (diameter)
C	End angle—for angle direction, see graphic support window
XK	End point (Cartesian)
YK	End point (Cartesian)
Z	End point (default: current Z position)

Programming:

- **X, C, XK, YK, Z:** Absolute, incremental or modal
- Program either X–C or XK–YK

Danger of collision!

During G100 the tool moves on a linear path. To position the workpiece to a defined angle, use G110.



Example: G100

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N6 G100 XK20 YK5   [rapid traverse on front face]
N7 G101 XK50
N8 G103 XK5 YK50 R50
N9 G101 XK5 YK20
N10 G102 XK20 YK5 R20
N11 G14
N12 M15
. . .
```

Linear segment on front/rear face G101

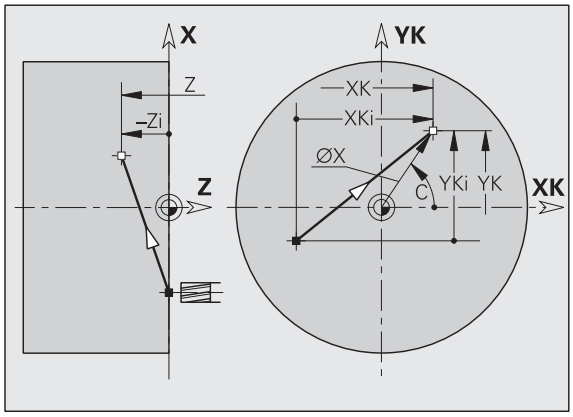
G101 moves the tool on a linear path at the feed rate to the "end point."

Parameters

X	End point (diameter)
C	End angle—for angle direction, see help graphic
XK	End point (Cartesian)
YK	End point (Cartesian)
Z	End point (default: current Z position)

Programming:

- **X, C, XK, YK, Z:** Absolute, incremental or modal
- Program either X–C or XK–YK



Example: G101

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G100 XK50 YK0
N6 G1 Z-5
N7 G42 Q1
N8 G101 XK40    [linear path on front face]
N9 G101 YK30
N10 G103 XK30 YK40 R10
N11 G101 XK-30
N12 G103 XK-40 YK30 R10
N13 G101 YK-30
N14 G103 XK-30 YK-40 R10
N15 G101 XK30
N16 G103 XK40 YK-30 R10
N17 G101 YK0
N18 G100 XK110 G40
N19 G0 X120 Z50
N20 M15
. . .
```

Circular arc on front/rear face G102/G103

G102/G103 moves the tool in a circular arc at the feed rate to the "end point." The direction of rotation is shown in the graphic support window.

Parameters

X	End point (diameter)
C	End angle—for angle direction, see help graphic
XK	End point (Cartesian)
YK	End point (Cartesian)
R	Radius
I	Center point (Cartesian)
K	Center point (Cartesian)
Z	End point (default: current Z position)
H	Circular plane (working plane)—(default: 0)

- H=0, 1: Machining in XY plane (front face)
- H=2: Machining in YZ plane
- H=3: Machining in XZ plane

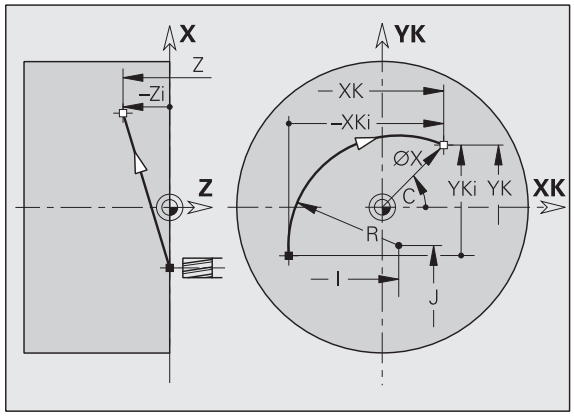
K	Center point for H=2, 3 (Z direction)

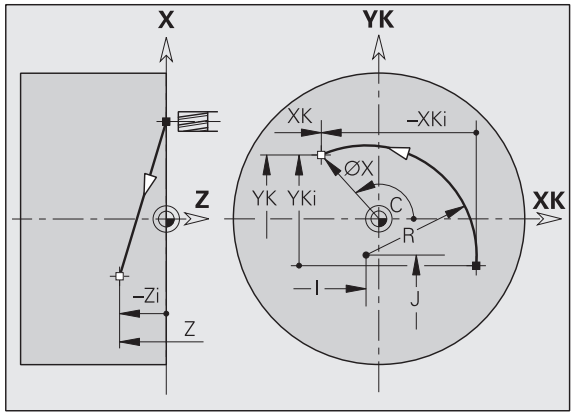
If you program H=2 or H=3, you can machine linear slots with a circular base. If

- H=2: Define the circle center with I and K.
- H=3: Define the circle center with J and K.

Programming:

- **X, C, XK, YK, Z:** Absolute, incremental or modal
- **I, J, K:** Absolute or incremental
- Program either X–C or XK–YK
- Program either center or radius
- For radius: Only arcs <= 180° are possible
- End point in the coordinate origin: Program XK=0 and YK=0.





Example: G102, G103

```
. . .
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N6 G100 XK20 YK5
N7 G101 XK50
N8 G103 XK5 YK50 R50          [circular arc]
N9 G101 XK5 YK20
N10 G102 XK20 YK5 R20
N12 M15
. . .
```

4.26 Lateral Surface Machining

Rapid traverse, lateral surface G110

G110 moves at rapid traverse along the shortest path to the end point.

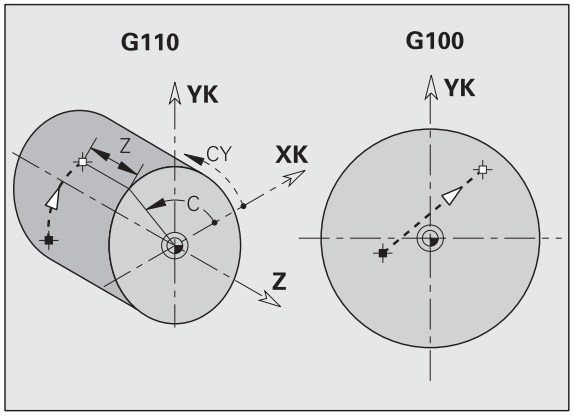
G110 is recommneded for **positioning the C axis** to a defined angle (programming: N.. G110 C...).

Parameters

Z	End point
C	End angle
CY	End point as linear value (referenced to unrolled reference diameter G120)
X	End point (diameter)

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- Program either Z–C or Z–CY



Example: G110

```
. . .
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0          [rapid, lateral surface]
N5 G0 X110 Z5
N6 G110 Z-20 CY0
N7 G111 Z-40
N8 G113 CY39.2699 K-40 J19.635
N9 G111 Z-20
N10 G113 CY0 K-20 J19.635
N11 M15
. . .
```

Line segment on lateral surface G111

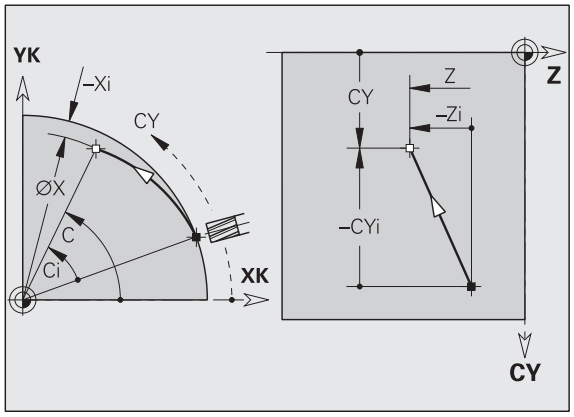
G111 moves the tool on a linear path at the feed rate to the "end point."

Parameters

Z	End point
C	End angle—for angle direction, see help graphic
CY	End point as linear value (referenced to unrolled reference diameter G120)
X	End point (diameter value)—(default: current X position)

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- Program either Z–C or Z–CY



Example: G111

```
. . .
[G111, G120]
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40        [linear path on lateral]
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
. . .
```

Circular arc on lateral surface G112/G113

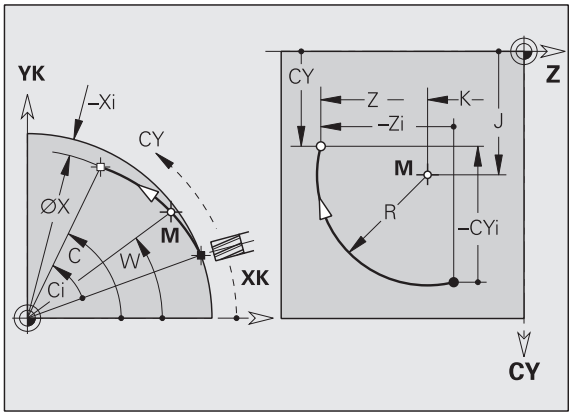
G112/G113 moves the tool in a circular arc at the feed rate to the "end point."

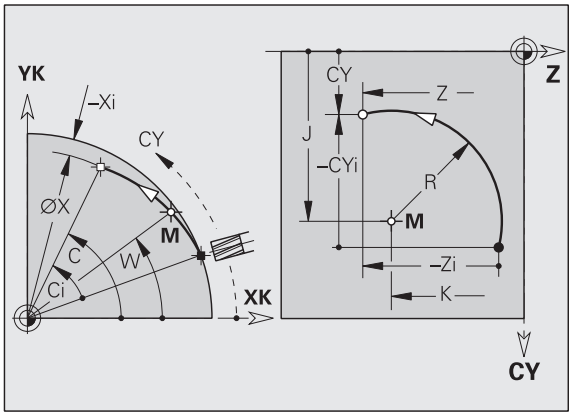
Parameters

Z	End point
C	End angle—for angle direction, see graphic support window
CY	End point as linear value (referenced to unrolled reference diameter G120)
R	Radius
K	Center
W	Center of angle (angular direction: see help graphic)
J	Center point as linear value (referenced to unrolled G120 reference diameter)
X	End point (diameter value)—(default: current X position)

Programming:

- **Z, C, CY:** Absolute, incremental, or modal
- **K; W, J:** Absolute or incremental
- Program either Z–C **or** Z–CY **and** K–J
- Program either center or radius
- For radius: Only arcs <= 180° are possible





Example: G112, G113

```
. . .
N1 T8 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635    [circular arc]
N10 G111 Z-20
N11 G112 CY0 K-20 J19.635
N13 M15
```

4.27 Milling Cycles

Contour milling G840—Fundamentals

G840 mills or deburrs open or closed contours (figures or "free contours"). Depending on the cutter, select vertical plunging or predrilling and then milling.

Plunge strategies: Depending on the cutter you are using, select one of the following strategies:

- **Vertical plunge:** The cycle moves the tool to the starting point; the tool plunges and mills the contour.
- **Calculate positions, predrill, mill.** The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with "G840 A1 ..".
  - Predrill with "G71 NF .."
  - Call cycle "G840 A0 ..". The cycle positions the tool above the hole; the tool plunges and mills the contour.
- **Predrilling, milling.** The machining process is performed in the following steps:
  - Drill holes with "G71 .."
  - Position the cutter above the hole. Call cycle "G840 A0 ..". The cycle plunges and mills the contour or contour section.

If the pocket consists of multiple milling contours, G840 takes all the sections of the contour into account for drilling and milling. Call "G840 A0 .." separately for each section when calculating the hole positions without "G840 A1 ..".

Oversize: A G58 oversize "shifts" the contour to be milled in the direction given in "cycle type."

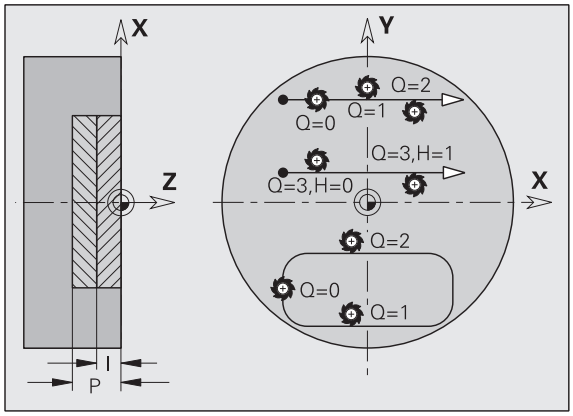
- With inside milling and closed contour: Shifted inward
- With outside milling and closed contour: Shifted outward
- Open contour: Shifts to the left or right depending on Q



- If Q=0, oversizes are not taken into account.
- G57 and negative G58 oversizes are not taken into account.

G840 – Calculating hole positions

"G840 A1 .." calculates the hole positions and stores them at the reference specified in "NF." Program only the parameters given in the following list.



See also:

- G840—Fundamentals: Page 261
- G840—Milling: Page 263

Parameters – Calculating hole positions

Q Cycle type (= milling location)

- Open contour. If there is any overlapping, Q defines whether the first area (as of starting point) or the entire contour is to be machined.
  - Q=0: Milling center on the contour (hole position = starting point)
  - Q=1: Machining at the left of the contour. If there is overlapping, only the first area of the contour is machined.
  - Q=2: Machining at the right of the contour. If there is overlapping, only the first area of the contour is machined.
  - Q=3: Not allowed
  - Q=4: Machining at the left of the contour. If there is overlapping, the entire contour is machined.
  - Q=5: Machining at the right of the contour. If there is overlapping, the entire contour is machined.
- Closed contours
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Inside milling
  - Q=2: Outside milling
  - Q=3 to 5: Not allowed

NS Block number – beginning of contour section

- Figures: Block number of the figure
- Free closed contour: First contour element (not starting point)
- Open contour: First contour element (not starting point). NS – NE defines the contour direction.

NE Block number – end of contour section

- Figures, free closed contour: No input
- Exposed contour: last contour element
- Contour consists of an element:
  - No input: Machining in contour direction
  - NS=NE programmed: Machining against the contour direction

Parameters – Calculating hole positions

D Starting element number for partial figures

The direction of contour definition for figures is counterclockwise. The first contour element for figures:

- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The position angle points to the first contour element.

V Ending element number for partial figures

A Sequence for "Calculate hole positions": A=1

NF Position mark – reference at which the cycle stores the hole positions [1..127].

WB Rework diameter—diameter of the milling cutter

Program D and V to machine parts of a figure.

- The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling "G840 A1 ..".
- Program oversizes for calculating the hole positions **and** for milling.

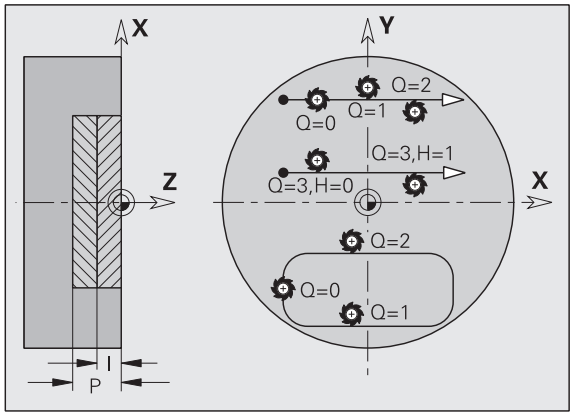
G840 overwrites any hole positions that may still be stored at the reference "NF."

G840 – Milling

You can change the machining direction and the cutter radius compensation (TRC) with the cycle type Q, the cutting direction H and the rotational direction of the tool (see following table). Program only the parameters given in the following list.

See also:

- G840—Fundamentals: Page 261
- G840—Calculating hole positions: Page 262



Parameters – milling

Q Cycle type (= milling location).

- Open contour. If there is any overlapping, Q defines whether the first area (as of starting point) or the entire contour is to be machined.
  - Q=0: Milling center on the contour (without TRC)
  - Q=1: Machining at the left of the contour. If there is overlapping, G840 machines only the first area of the contour.
  - Q=2: Machining at the right of the contour. If there is overlapping, G840 machines only the first area of the contour.
  - Q=3: The contour is machined to the left or right depending on H and the direction of cutter rotation (see table). If there is overlapping, G840 machines only the first area of the contour.
  - Q=4: Machining at the left of the contour. If there is overlapping, G840 machines the entire contour.
  - Q=5: Machining at the right of the contour. If there is overlapping, G840 machines the entire contour.
- Closed contours
  - Q=0: Center of milling cutter on the contour (hole position = starting point)
  - Q=1: Inside milling
  - Q=2: Outside milling
  - Q=3 to 5: Not allowed

NS Block number—beginning of contour section

- Figures: Block number of the figure
- Free open or closed contour: First contour element (not starting point)

NE Block number—end of contour section

- Figures, free closed contour: No input
- Free open contour: Last contour element
- Contour consists of one element:
  - No input: Machining in contour direction
  - NE programmed: Machining against the contour direction

H Cutting direction (default: 0)

- H=0: Up-cut milling
- H=1: Climb milling

I (Maximum) infeed (default: milling in one infeed)

F Infeed rate (depth infeed) – (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

Parameters – milling

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; feed to starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
- R<0 for inside corners: Tool moves on approaching/ departing arc that connects tangentially to the contour element.
- R<0 for outside corners: Contour element is approached/ departed tangentially on a line

P Milling depth (default: depth from the contour description)

K Retraction plane (default: back to starting position)

- Front or rear face: Retraction position in Z direction
- Lateral surface: Retraction position in X direction (diameter)

D Starting element number when partial figures are machined.

V End element number when partial figures are machined.

The direction of contour definition for figures is counterclockwise. The first contour element for figures:

- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The orientation angle points to the first contour element.

A Sequence for "Milling, deburring": A=0 (default=0)

NF Position mark—reference from which the cycle reads the hole positions [1 to 127].

O Approach behavior (default: 0)

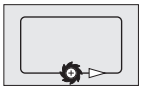
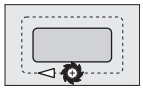
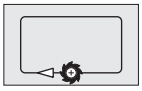
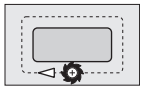
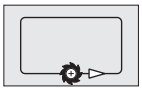
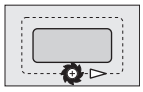
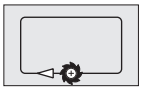
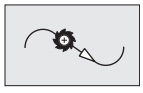
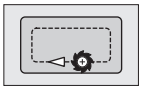
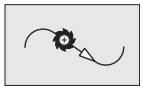
- O=0: Vertical plunging
- O=1: With predrilling
  - If NF is programmed: The cycle positions the milling cutter above the first hole position saved in NF, then plunges and mills the first section. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next section, etc.
  - If NF is not programmed: The tool plunges at the current position and mills the section. If required, repeat this operation for the next section, etc.

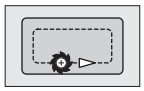
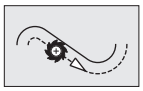
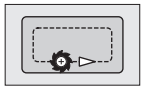
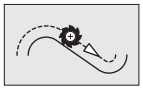
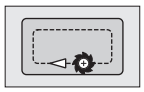
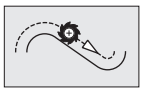
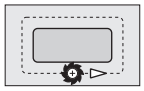
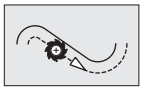
Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use D and V to select the approach/depart.

Cycle run for milling

1. Starting position (X, Z, C) is the position before the cycle begins.
2. Calculates the milling depth infeeds.
3. Approaches to safety clearance.
   - If O=0: Infeed to the first milling depth.
   - If O=1: Rapid to the first milling depth.
4. Mills the contour.
5. - For open contour and slots with slot width equal to the cutter diameter: Moves at feed rate to the next milling depth, or plunges at rapid to the next milling depth and mills the contour in reverse direction.
   - For closed contours and slots: Lifts off by the safety clearance, moves forward and approaches to the next milling depth, or plunges to the next milling depth.
6. Repeats steps 4 and 5 until the complete contour is milled.
7. Return to "retraction plane K."

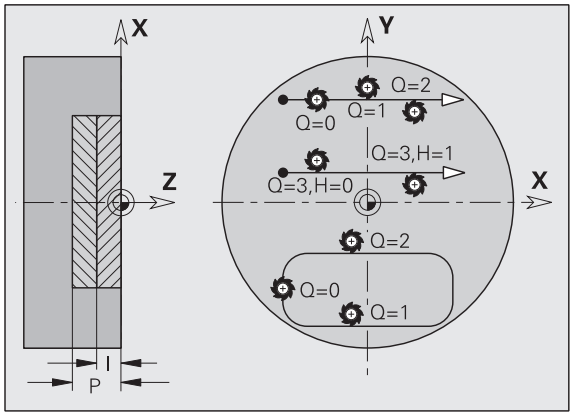
Contour milling G840

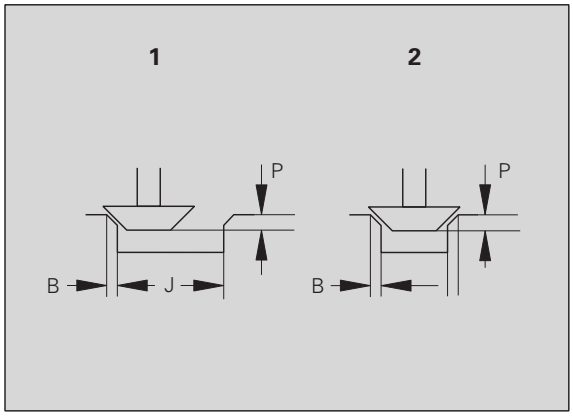
Cycle type	Cutting direction	Direction of tool rotation	TRC	Execution	Cycle type	Cutting direction	Direction of tool rotation	TRC	Execution
Contour (Q=0)	–	Mx03	–		Outside	Up-cut milling (H=0)	Mx04	Left	
Contour	–	Mx03	–		Outside	Climb milling (H=1)	Mx03	Left	
Contour	–	Mx04	–		Outside	Climb milling (H=1)	Mx04	Right	
Contour	–	Mx04	–		Contour (Q=0)	–	Mx03	–	
Inside (Q=1)	Up-cut milling (H=0)	Mx03	Right		Contour	–	Mx04	–	

Contour milling G840									
Cycle type	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Execution**	**Cycle type**	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Execution**
Inside	Up-cut milling (H=0)	Mx04	Left		Right (Q=3)	Up-cut milling (H=0)	Mx03	Right	
Inside	Climb milling (H=1)	Mx03	Left		Left (Q=3)	Up-cut milling (H=0)	Mx04	Left	
Inside	Climb milling (H=1)	Mx04	Right		Left (Q=3)	Climb milling (H=1)	Mx03	Left	
Outside (Q=2)	Up-cut milling (H=0)	Mx03	Right		Right (Q=3)	Climb milling (H=1)	Mx04	Right	

G840 – Deburring

G840 deburrs when you program chamfer width B. If there is any overlapping of the contour, specify with Q whether the first area (as of starting point) or the entire contour is to be machined. Program only the parameters given in the following list.





Parameters – deburring

Q Cycle type (= milling location)

- Open contour
  - Q=0: Milling center on the contour. Q0 deburrs the slot in one pass on the previously open or closed contour.
  - Q=1: Machining at the left of the contour. If there is overlapping, G840 machines only the first area of the contour.
  - Q=2: Machining at the right of the contour. If there is overlapping, G840 machines only the first area of the contour.
  - Q=3: The contour is machined to the left or right depending on H and the direction of cutter rotation (see "G840 – Milling" on page 263). If there is overlapping, G840 machines only the first area of the contour.
  - Q=4: Machining at the left of the contour. If there is overlapping, G840 machines the entire contour.
  - Q=5: Machining at the right of the contour. If there is overlapping, G840 machines the entire contour.
- Closed contours
  - Q=0: Milling center on the contour
  - Q=1: Inside milling
  - Q=2: Outside milling

NS Block number—beginning of contour section

- Figures: Block number of the figure
- Free open or closed contour: First contour element (not starting point)

NE Block number—end of contour section

- Figures, free closed contour: No input
- Free open contour: Last contour element
- Contour consists of one element:
  - No input: Machining in contour direction
  - NE programmed: Machining against the contour direction

E Reduced feed rate for circular elements (default: current feed rate)

Parameters – deburring

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
- R<0 for inside corners: Tool moves on approaching/ departing arc that connects tangentially to the contour element.
- R<0 for outside corners: Contour element is approached/ departed tangentially on a line

P Milling depth. Plunging depth of the tool

K Retraction plane (default: back to starting position)

- Front or rear face: Retraction position in Z direction
- Lateral surface: Retraction position in X direction (diameter)

B Chamfer width when deburring the edges

J Preparation diameter For open contours, the contour to be deburred is calculated from the programmed contour and J.

The following rules apply:

- J programmed: The cycle deburrs both sides of the slot (see 1 in the illustration).
- J not programmed: The deburring tool is so wide that both sides of the slot are deburred in one pass (see 2 in the illustration).

D Starting element number when partial figures are machined.

V End element number when partial figures are machined.

The direction of contour definition for figures is counterclockwise. The first contour element for figures:

- Circular slot: The larger arc
- Full circle: The upper semicircle
- Rectangles, polygons and linear slots: The orientation angle points to the first contour element.

A Sequence for "Milling, deburring": A=0 (default=0)

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use D and V to select the approach/departure element.

Cycle run for deburring

1. Starting position (X, Z, C) is the position before the cycle begins.
2. Moves to the safety clearance and infeed to the first milling depth.
3. ■ J not programmed: Mills the programmed contour.
   ■ J programmed, open contour: Calculate and mills the "new" contour.
4. Return to "retraction plane K."

Pocket milling, roughing G845 – Fundamentals

G845 roughs closed contours. Choose one of the following **plunge strategies**, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When "plunging at a pre-drilled position," you have the following alternatives:

- **Calculate positions, drill, mill.** The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with "G845 A1 ..".
  - Predrill with "G71 NF .."
  - Call cycle "G845 A0 ..". The cycle positions the tool above the hole; the tool plunges and mills the pocket.
- **Drill, mill.** The machining process is performed in the following steps:
  - Drill a hole inside the pocket with "G71 ..".
  - Position the milling cutter above the hole and call "G845 A0 ..". The tool plunges and mills the section.

If the pocket consists of multiple sections, G845 takes all the sections of the pocket into account for drilling and milling. Call "G845 A0 .." separately for each section when calculating the hole positions without "G845 A1 ..".



G845 takes the following oversizes into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Program oversizes for calculating the hole positions **and** for milling.

G845 – Calculating hole positions

"G845 A1 .." calculates the hole positions and stores them at the reference specified in "NF." The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling "G845 A1 ..". Program only the parameters given in the following list.

See also:

- G845—Fundamentals: Page 270
- G845—Milling: Page 272

Parameters—Calculating hole positions

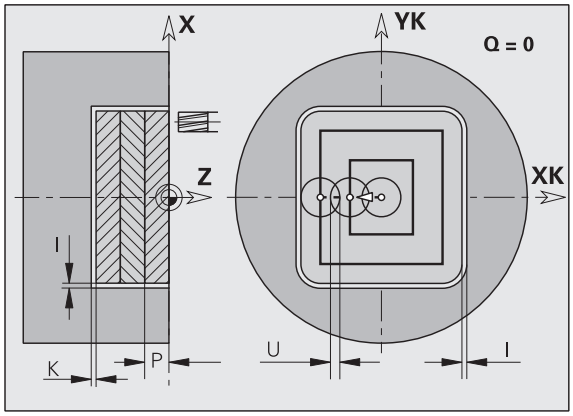
NS	Block number – reference to the contour description
I	Oversize in X direction
K	Oversize in Z direction
Q	Machining direction (default: 0) ■ Q=0: From the inside toward the outside ■ Q=1: From the outside toward the inside
A	Sequence for "Calculate hole positions": A=1
NF	Position mark—reference at which the cycle stores the hole positions [1 to 127].
WB	Plunge length – diameter of the milling cutter

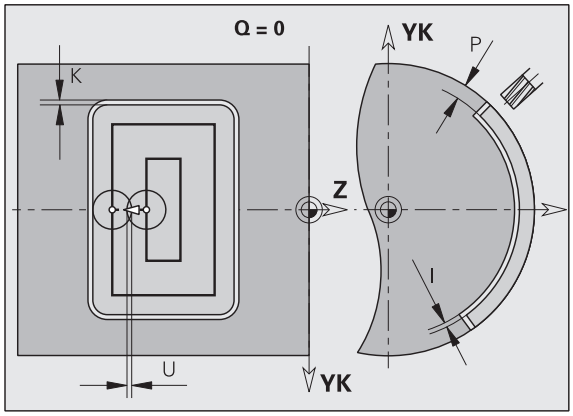


- G845 overwrites any hole positions that may still be stored at the reference "NF."
- The parameter "WB" is used both for calculating the hole positions and for milling. When calculating the hole positions, "WB" describes the diameter of the milling cutter.

G845 – Milling

You can change the **milling direction** with the cutting direction H, the machining direction Q and the direction of tool rotation (see following table). Program only the parameters given in the following list.





See also:

- G845—Fundamentals: Page 270
- G845—Calculating hole positions: Page 271

Parameters—Milling

NS Block number—reference to the contour description

P (Maximum) milling depth (infeed in the working plane)

I Oversize in X direction

K Oversize in Z direction

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

H Cutting direction (default: 0)

- H=0: Up-cut milling
- H=1: Climb milling

F Feed rate for infeed (default: active feed rate)

Starting with software version 625 952-05: F is used as machining feed rate for reciprocating or helical plunging.

E Reduced feed rate for circular elements (default: current feed rate)

J Retraction plane (default: back to starting position)

- Front or rear face: Retraction position in Z direction
- Lateral surface: Retraction position in X direction (diameter)

Q Machining direction (default: 0)

- Q=0: From the inside toward the outside
- Q=1: From the outside toward the inside

A Sequence for “Milling”: A=0 (default=0)

NF Position mark—reference from which the cycle reads the hole positions [1 to 127].

O Plunging behavior (default: 0)

Plunge vertically O=0: The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket.

Parameters—Milling

Plunge at pre-drilled position O=1:

- If "NF" is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
- If "NF" is not programmed: The tool plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.

Plunge in a helical motion O=2, 3: The tool plunges at the angle "W" and mills full circles with the diameter "WB." As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=2 – manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=3 – automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends on the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas.

Plunge in a reciprocating linear motion O=4, 5: The tool plunges at the angle "W" and mills a linear path of the length "WB." You can define the position angle in "WE." The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=4 – manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=5 – automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends on the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from "Q:"
  - Q0 (from the inside toward the outside):
    – Linear slot, rectangle, polygon: Reference point of the figure
    – Circle: Circle center
    – Circular slot, "free" contour: Starting point of the innermost milling path
  - Q1 (from the outside toward the inside):
    – Linear slot: Starting point of the slot
    – Circular slot, circle: Not machined
    – Rectangle, polygon: Starting point of the first linear element
    – "Free" contour: Starting point of the first linear element (at least one linear element must exist)

Parameters—Milling

Plunge in a reciprocating circular motion O=6, 7: The tool plunges at the plunging angle "W" and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling. "WE" defines the arc center, "WB" the arc radius.

- O=6 – manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges.
- O=7 – automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of "Q:"
  - Q0 (from the inside toward the outside):
    – Circular slot: The circular arc lies on the curvature radius of the slot
    – Circle: Not permitted
  - Q1 (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path

W Plunging angle in infeed direction

WE Position angle of the milling path/circular arc. Reference axis:

- Face or back: Positive XK axis
- Lateral surface: Positive Z axis

Default position angle, depending on "O:"

- O=4: WE= 0°
- O=5 and
  - Linear slot, rectangle, polygon: WE= position angle of the figure
  - Circular slot, circle: WE=0°
  - "Free" contour and Q0 (from the inside toward the outside): WE=0°
  - "Free" contour and Q1 (from the outside toward the inside): Position angle of the starting element

WB Plunge length/plunge diameter (default: 1.5 * milling diameter)



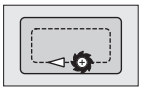
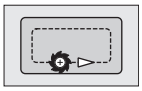
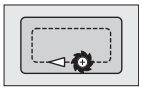
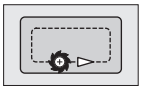
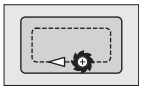
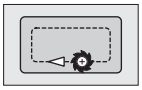
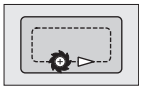
For the machining direction Q=1 (from the outside toward the inside), please note:

- The contour must start with a linear element.
- If the starting element is < WB, WB is reduced to the length of the starting element.
- The length of the starting element must not be less than 1.5 times the diameter of the milling cutter.

Cycle run

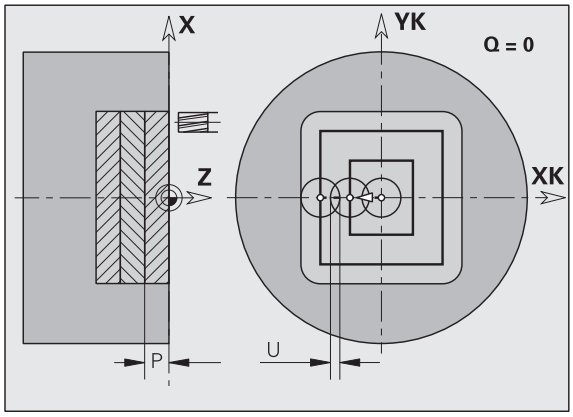
1. Starting position (X, Z, C) is the position before the cycle begins.
2. Calculates the number of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging positions and paths for reciprocating or helical plunges.
3. Approaches to safety clearance and, depending on O, feeds to the first milling depth or approaches helically or on a reciprocating path, .
4. Mills a plane.
5. Retracts by the safety clearance, returns and cuts to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to retraction plane J.

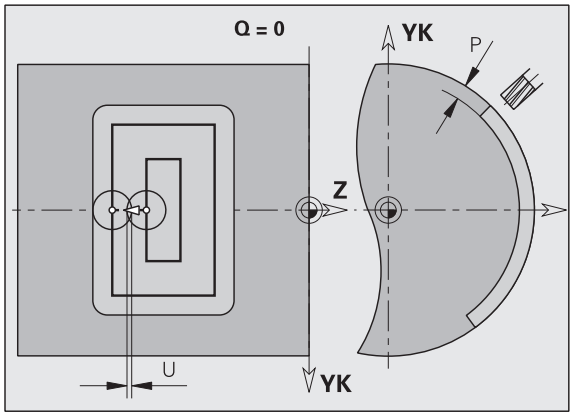
Pocket milling, roughing G845

Cutting direction	Machining direction	Direction of tool rotation	Execution	Cutting direction	Machining direction	Direction of tool rotation	Execution
Up-cut milling (H=0)	From inside (Q=0)	Mx03		Climb milling (H=1)	From inside (Q=0)	Mx03	
Up-cut milling (H=0)	From inside (Q=0)	Mx04		Climb milling (H=1)	From inside (Q=0)	Mx04	
Up-cut milling (H=0)	From outside (Q=1)	Mx03		Climb milling (H=1)	From outside (Q=1)	Mx03	
Up-cut milling (H=0)	From outside (Q=1)	Mx04		Climb milling (H=1)	From outside (Q=1)	Mx04	

Pocket milling, finishing G846

You can change the **milling direction** with the cutting direction H, the machining direction Q and the direction of tool rotation (see following table).





Machining parameters – finishing

NS Block number – reference to contour description

P (Maximum) milling depth (infeed in the working plane)

R Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly. Feed to the starting point above the milling plane, then vertical plunge.
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.

U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).

Overlap = U*milling diameter

H Cutting direction (default: 0)

- H=0: Up-cut milling
- H=1: Climb milling

F Approach feed for infeed (default: active feed rate)

E Reduced feed rate for circular elements (default: current feed rate)

J Retraction plane (default: back to starting position)

- Front or rear face: Retraction position in Z direction
- Lateral surface: Retraction position in X direction (diameter)

Q Machining direction (default: 0)

- Q=0: From the inside toward the outside
- Q=1: From the outside toward the inside

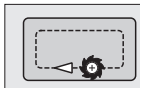
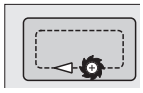
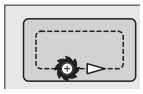
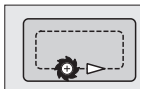
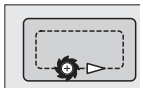
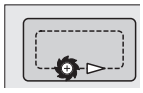
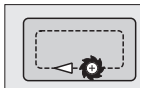
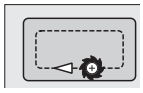
O Plunging behavior (default: 0)

- O=0 – plunge vertically: The cycle moves the tool to the starting point; the tool plunges and finishes the pocket.
- Q=1 – Approaching arc with depth feed: When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc "R" and when machining from the outside toward the inside (Q=1).

Cycle run

1. Starting position (X, Z, C) is the position before the cycle begins.
2. Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths).
3. Moves to the safety clearance and feeds to the first milling depth.
4. Mills a plane.
5. Retracts by the safety clearance, returns and cuts to the next milling depth.
6. Repeat steps 4 and 5 until the complete surface is milled.
7. Returns to retraction plane J.

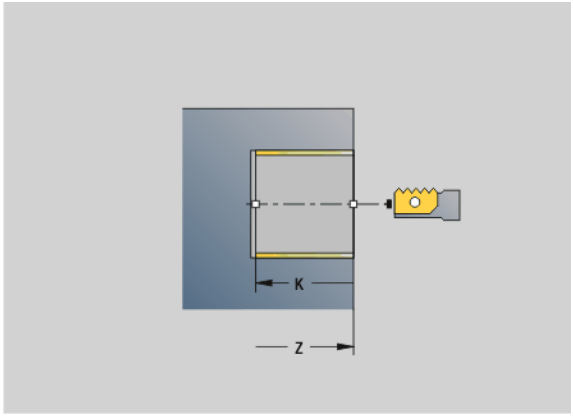
Pocket milling, finishing G846

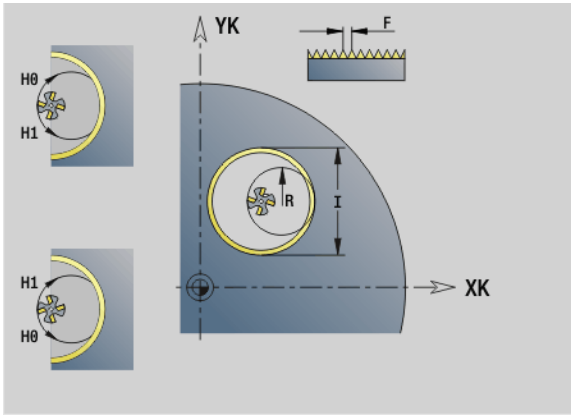
Cutting direction	Direction of tool rotation	Execution	Cutting direction	Direction of tool rotation	Execution
Up-cut milling (H=0)	Mx03		Up-cut milling (H=0)	Mx03	
Up-cut milling (H=0)	Mx04		Up-cut milling (H=0)	Mx04	
Climb milling (H=1)	Mx03		Climb milling (H=1)	Mx03	
Climb milling (H=1)	Mx04		Climb milling (H=1)	Mx04	

Thread milling, axial G799

Starting with software version 625 952-05: G799 mills a thread in existing holes.

The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one revolution or, with single-point tools, in several revolutions.





Parameters

X	Polar starting point
C	Polar starting point
XK	Cartesian starting point
YK	Cartesian starting point
Z	Milling top edge
I	Thread diameter
K	Thread depth
R	Approach radius
F	Thread pitch
J	Direction of thread—(default: 0) ■ 0: Right-hand thread ■ 1: Left-hand thread
H	Cutting direction (default: 0) ■ 0: Up-cut milling ■ 1: Climb milling
V	One rotation / several rotations ■ 0: The thread is milled in a 360-degree rotation ■ 1: The thread is milled in several rotations (single-point tool)



Use thread-milling tools for cycle G799.

Danger of collision!

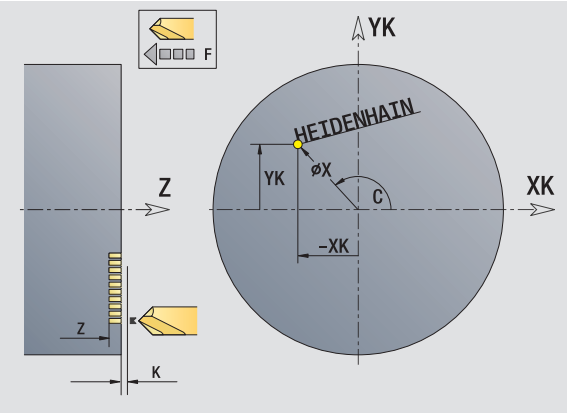
The hole depth must exceed the thread depth by at least F/2.

Example: G799

```
%799.nc
[G799]
N1 T9 G195 F0.2 G197 S800
N2 G0 X100 Z2
N3 M14
N4 G799 XK100 C45 Z0 I12 K-20 F2 J0 H0 V0
N5 M15
END
```

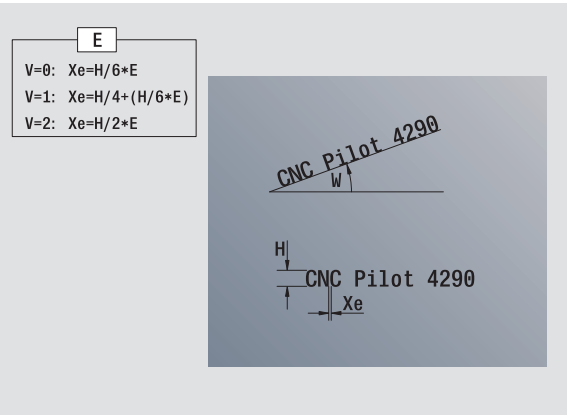
Engraving on front face G801

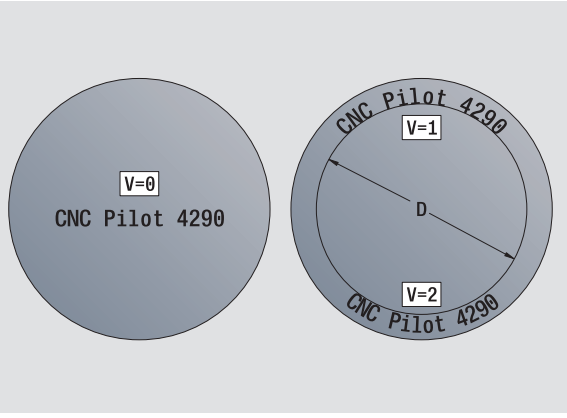
G801 engraves character strings in linear or polar layout on the front face. The text to be engraved is entered in the "ID" box as a character string.



Parameters

ID	Text. Text to be engraved ()
NS	Character number. ASCII code of the character to be engraved
X	Starting diameter (polar coordinates)
C	Starting angle (polar coordinates)
XK	Starting point in Cartesian coordinates
YK	Starting point in Cartesian coordinates
Z	Milling floor. Z position, infeed depth during milling.
K	Retraction plane. Z position retracted to for positioning.
H	Font height. Height of the characters in [mm]
W	Position angle of the character string with linear alignment. Example: 0° = Vertical characters: the characters are aligned in sequence in positive XK direction
E	Distance factor (default: 1). The distance between the characters is calculated depending on V: ■ V=0: Distance = H/6 * E ■ V=1: Distance = H/4 + (H/6 * E) ■ V=2: Distance = H/2 * E
V	Linear/polar execution (default: 0) ■ V=0: Characters are aligned linearly ■ V=1: Characters are bent upward about the center ■ V=2: Characters are bent downward about the center
D	Reference diameter for polar design
F	Available as of software version 625 952-05. Approach feed factor (feed rate = current feed rate * F)





Diacritics and special characters that you cannot enter in the DIN editor can be defined, character by character, in NS. If text is defined in ID and a character is defined in NS, the text is engraved before the character.

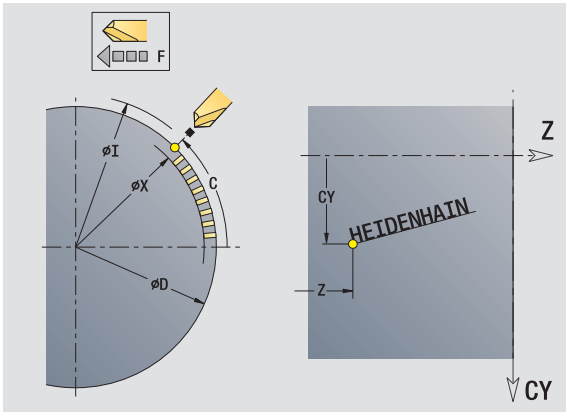
G801 starts engraving from the starting position, or from the current position if no starting position is defined.

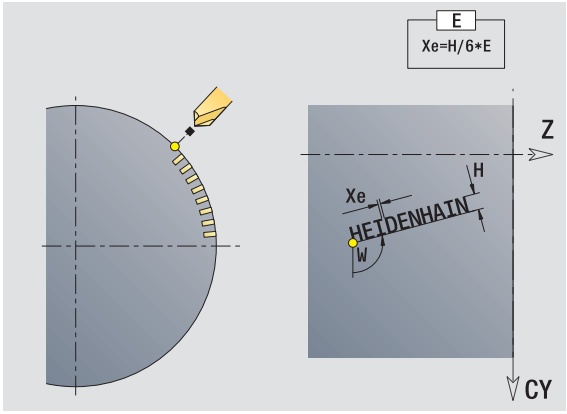
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Character set: see "Character set for engraving" on page 280

Engraving on lateral surface G802

G802 engraves character strings aligned linearly on the lateral surface. The text to be engraved is entered in the "ID" box as a character string.





Parameters

ID	Text. Text to be engraved ()
NS	Character number. ASCII code of the character to be engraved
Z	Starting point
C	Starting angle
CY	Starting angle as line dimension (reference: unrolled reference diameter)
X	Milling diameter. X position, infeed depth during milling.
I	Retraction diameter. X position retracted to for positioning.
H	Font height. Height of the characters in [mm]
W	Position angle of the character string. Examples: ■ 0°: from –CY to +CY ■ 90°: from –Z to +Z
E	Distance factor (default: 1). The distance between the characters is calculated according to the following formula: H / 6 * E
D	Reference diameter for calculating the line dimension CY
F	Available as of software version 625 952-05. Approach feed factor (feed rate = current feed rate * F)

Diacritics and special characters that you cannot enter in the DIN editor can be defined, character by character, in NS. If text is defined in ID and a character is defined in NS, the text is engraved before the character.

G802 starts engraving from the starting position, or from the current position if no starting position is defined.

Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Character set: see "Character set for engraving" on page 280

Character set for engraving

The CNC PILOT can realize the characters listed in the following table. Enter the character code NS if the character to be engraved cannot be depicted in the DIN editor.

Small letters		Capital letters		Numerals, diacritics		Special characters		
NS	**Character**	**NS**	**Character**	**NS**	**Character**	**NS**	**Character**	**Meaning**
97	a	65	A	48	0	32		Blank space
98	b	66	B	49	1	37	%	Per cent sign
99	c	67	C	50	2	40	(	Opening parenthesis
100	d	68	D	51	3	41	)	Closing parenthesis
101	e	69	E	52	4	43	+	Plus character
102	f	70	F	53	5	44	,	Comma
103	g	71	G	54	6	45	–	Minus sign
104	h	72	H	55	7	46	.	Point
105	i	73	I	56	8	47	/	Forward slash
106	j	74	J	57	9	58	:	Colon
107	k	75	K			60	<	Less than character
108	l	76	L	196	Ä	61	=	Equal sign
109	m	77	N	214	Ö	62	>	Greater than character
110	n	78	N	220	Ü	64	@	at
111	o	79	O	223	ß	91	[	Opening brackets
112	p	80	P	228	ä	93	]	Closing brackets
113	q	81	Q	246	ö	95	_	Underscore
114	r	82	R	252	ü	128	?	Euro sign
115	s	83	S			181	µ	Micron symbol
116	t	84	T			186	°	Degrees
117	u	85	U			215	x	Multiplication sign
118	v	86	V			Available as of software version 625 952-05.		
119	w	87	W			33	!	Exclamation point
120	x	88	X			38	&	Ampersand and
121	y	89	Y			63	?	Question mark
122	z	90	Z			174	®	Trademark
						216	Ø	Diameter sign

4.28 Assignment, Synchronization, Workpiece Transfer

Multichannel systems

The CNC PILOT controls one slide per NC channel. Lathes with two or more slides are referred to as multichannel systems.

Examples:

- Machines for full-surface machining with opposing spindles
- Multiple slide working on one workpiece
- Two or more workpieces machine in one working space.

Such operations are programmed in **one** NC program. The challenge for the NC programmer is in optimally distributing the operations among several slides/spindles and synchronizing them correctly. The CNC PILOT support that effort by:

- Organizational instructions (assignments of contours/program section to slides/spindles, etc.)
- Synchronization commands
- Mirroring of contours, tool dimensions and paths of traverse
- Conversion of G and M functions

Converting and mirroring G30

G30 converts G functions, M functions, and slide and spindle numbers with the aid of the conversion lists (MP 135 and following). G30 mirrors traverse paths and tool dimensions and shifts the machine zero point about the "zero point offset" of the axis (MP 1114, 1164, ..).

Parameters

H — Table number of the conversion table

- H=0: Switch off conversion and compensate the offset
- H=1 to 4: Activate conversion table 1 to 4 and machine zero point shift (MP 1114, 1164, ...)

Q — Selection. Switch-on/switch-off traverse-path mirroring/tool mirroring for given axes

- Q=0: Switch-off traverse-path milling and tool mirroring
- Q=1: Switch-on traverse-path mirroring
- Q=2: Switch-on tool-dimension mirroring

X, Y, Z, ... — Axis mirroring on/off

- X=0: X-axis mirroring off
- X=1: X-axis mirroring on
- Y=0: Y-axis mirroring off
- Y=1: Y-axis mirroring on
- . . .

Application: For full-surface machining, you describe the complete contour, machine the front face, rechuck the workpiece using an expert program, and then machine the rear face. To enable you to program rear-face machining in the same way as front-face machining (Z-axis orientation, arc rotational direction, etc.). Includes the expert program commands for converting and mirroring.



- Mirror the traverse paths **and** tool lengths in separate G30 commands.
- Q1, Q2 without axis selection switches the mirroring function off.
- Only configured axes can be selected.



Danger of collision!

- In the transition from AUTOMATIC to MANUAL OPERATION, conversions and mirror images are retained.
- Switch off the conversion/mirroring if you activate the front-face machining after rear-face machining (for example during program section repeats with M99).
- After a new program selection, the conversion/mirroring is switched off (example: transition from MANUAL to AUTOMATIC mode).

Spindle with workpiece G98

The assignment of the spindle is required for thread-cutting, drilling, and milling cycles if the workpiece is not in the main spindle.

Parameters

Q	Spindle number (0 to 3); (default: 0 = main spindle)

Workpiece group G99

For two or more contours (workpieces) in one NC program, use **CONTOUR Q..** (see "CONTOUR section" on page 143). G99 assigns the "Contour Q" to the following operations. The slide code before the NC block defines the slides that machine this contour. If G99 was not yet programmed (for example at the start of the program), all slides on contour 1.

Parameters

Q	Workpiece number (1 to 4)—number of the contour
D	Spindle number (1 to 4)—spindle that holds the workpiece
X	Shift of X for the simulation (diameter)
Z	Shift of Z for the simulation



- Program G99 again if the workpiece is transferred to another spindle and/or moves its position in the working space.
- The simulation
  - positions the tool using the "shift X, Z,"
  - finds and positions the chucking equipment using the spindle number D (G99 does not replace G65).

One-sided synchronization G62

The slide programmed with G62 waits until slide Q has reached mark H or the mark and the X/Z coordinate. The mark sets another slide with G162.

Parameters

H	Number of the mark (range: 0 <= H <= 15)
Q	Slide to be awaited
X	Coordinate to end the waiting process (default: synchronization only to the mark)
Z	Coordinate to end the waiting process (default: synchronization only to the mark)



- Both slides must be controlled by the common main program.
- The CNC PILOT synchronizes to the actual value. Therefore, do not synchronize to the end coordinates of NC blocks, because due to servo lag the positions might not be reached.
- Alternative: Synchronous start of paths with G63

Example of machining with G62

`. . .`	
`$1 N.. G62 Q2 H5`	Slide $1 waits until slide $2 reaches the mark 5
`. . .`	
`$2 N.. G62 Q1 H7 X200`	Slide $2 waits until slide $1 reaches the mark 7 and the position X200
`. . .`	

Synchronization marking G162

G162 sets a synchronous mark. (Another slide programmed with G62 waits until the mark has been crossed.) The slide continues executing the NC program without a pause.

Parameters

H Number of the mark (0 <= H <= 15)

Synchronous start of slides G63

G63 causes a **synchronous (simultaneous) start** of the programmed slides.



Ensure that no M or T commands are contained between the NC block containing G63 and the blocks containing traverse commands.

Example: Synchronization with G63

```
. . .
[Slides $1, $2 start simultaneously]
$1 $2 N.. G63
$1 N.. G1 X.. Z..
$2 N.. G1 X.. Z..
. . .
```

M97 Synchronous function

Slides for which M97 is programmed wait until all slides have reached this block. Program run then continues.

For complex machining operations (e.g. machining of several workpieces), M97 is programmed with parameters.

Parameters

H	Synchronous mark number – the evaluation takes place only during interpretation of the NC programs
Q	Slide number – use synchronization with Q if synchronization with $x is not possible
D	On/Off (default: 0)

- 0: Off – Synchronization to run time of the NC program
- 1: On – synchronization only during interpretation of the NC programs

Example: Synchronization with M97

```
. . .
[Slides $1, $2 wait for each other]
$1 N.. G1 X.. Z..
$2 N.. G1 X.. Z..
$1$2 N.. M97
. . .
```

Spindle synchronization G720

G720 controls the workpiece transfer from the master to the slave spindle and synchronizes functions such as polygonal turning jobs.

Parameters

S	Number of the master spindle [1 to 4]
H	Number of the slave spindle [1 to 4]—no input or H=0: Switches off the spindle synchronization
C	Offset angle [°] (default: 0°)
Q	Master rotational speed factor (default: 1)
	Range: -100 <= Q <= 100
F	Master rotational speed factor (default: Q is loaded)
	Range: –100 <= F <= 100
J	Slave gear ratio

Program the speed of the master spindle with Gx97 S.. and define the speed ratio between the master spindle and the slave spindle with Q, F. If you enter a negative value for Q or F, the direction of rotation of the slave spindle will be reversed.

In the slave gear ratio J, enter the transmission ratio if the slave spindle is driven through a transmission.

Remember that: **Q * master speed = F * slave speed**

Example: G720

`. . .`	
`N.. G397 S1500 M3`	Speed of direction of rotation of master spindle
`N.. G720 C180 S4 H2 Q2 F-1`	Synchronization of master spindle and slave spindle. The slave spindle precedes the master spindle by 180°. Slave spindle: Direction of rotation M4; rotational speed 750
`$2 N.. G1 X.. Z..`	
`. . .`	
`. . .`	

C-angle offset G905

G905 measures the angular offset of workpiece transfer with rotating spindle. The sum of angle C and the angle offset goes into effect as the zero point shift of C axis. This value is saved in the variable V922 (C axis 1) or V923 (C axis 2).

The zero offset is effective internally as a direct zero point shift for the respective C axis. The content of the variables is retained even if the control has been switched off. The control does not initialize these values automatically. If required, initialize the values manually by overwriting the relevant variables.

Parameters

Q Number of the C axis

C Angle of additional zero point shift for offset gripping (–360° <= C <= 360°) – (default: 0°)



Danger of collision!

- For narrow workpieces the jaws have to grip at an offset.
- The zero point shift, C axis, remains in effect:
  - After switch from Automatic to Manual mode
  - After switch-off

Measuring angular offset during spindle synchronization G906

G906 writes the angular offset between the master spindle and the slave spindle into variable V921.

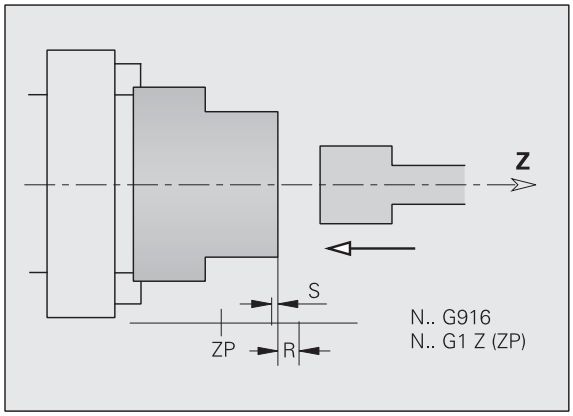
Programming:

- Program G906 only for active angular synchronization – both chucks must be closed.
- Program G906 in a separate NC block.
- Program a G909 (interpreter stop) **before processing** V921.
- G906 generates an interpreter stop

Traversing to a fixed stop G916

G916 activates the monitoring function for the traversing path. Then you move with G1 to a fixed stop. User G916 for:

- Traversing to a dead stop (example: transferring a premachined workpiece to a second traveling spindle if you do not know the exact position of the workpiece).
- Press the tailstock against the workpiece (tailstock function)



Parameters

H	Clamping force in daNewtons (1 daNewton = 10 newtons)
D	Mode: ■ D=1: Activate tailstock function. ■ D=2: Deactivate tailstock function. As of software version 625 952-04: ■ D=3: Do not issue message when end point is reached
R	Reversing path

The CNC PILOT stops the slide and saves the stop position. G916 generates an interpreter stop.

Traversing to a fixed stop (G916 without parameters). The CNC PILOT

- moves up to the fixed stop and stops as soon as the servo lag has been reached. The remaining path of traverse is deleted.
- saves the position of the fixed stop in variables V901 to V918.
- move back by the servo lag + reversing path (MP 1112, 1162, ..).

In MP 1112, 1162, .. you specify:

- Lag error limit
- Reversing path

Programming “traverse to a fixed stop”:

- Position the slide at a sufficient distance before the fixed stop.
- Use a moderate feed rate (< 1000 mm/min)
- Program G916 or G916 Hx D1 in the G1 positioning block
- Program G1 .. as follows:
  - Target position is behind the fixed stop
  - Move only **one** axis
  - Activate a per-minute feed rate (G94)

Example of traversing to a fixed stop:

`. . .`	
`$2 N.. G94 F200`	
`$2 N.. G0 Z20`	Pre-position slide 2
`$2 N.. G916 G1 Z-10`	Activate monitoring, traverse to a fixed stop
`. . .`	

Tailstock function (G916 with parameters)

- **G916 Hx D1** activates the tailstock function. The CNC PILOT
  - moves up to the workpiece and stops as soon as the contact force has been reached.
  - deletes the remaining path of traverse
- **G916 D2** deactivates the tailstock function. The CNC PILOT
  - deactivates the tailstock function.
  - moves back by the servo lag + reversing path (MP 1112, 1162, ..)

G916 D2 can be combined with a G1 positioning block

Example of tailstock function

`. . .`	
`$2 N.. G94 F800`	
`$2 N.. G0 Z20`	Pre-position slide 2
`$2 N.. G916 H250 D1 G1 Z-10`	Activate the tailstock function—contact force 250 daN
`. . .`	
`$2 N.. G916 D2 G1 Z100`	Deactivate the tailstock function and retract the tailstock

As of software version 625 952-04:

Check whether the end point is reached:

- **G916 D3**
  - On reaching the fixed stop, the CNC PILOT stops the slide and saves the stop position in the variables V901 to V918.
  - If the fixed stop is not reached, the CNC PILOT completes the programmed path of traverse and enters the error number 5519 in variable V982.

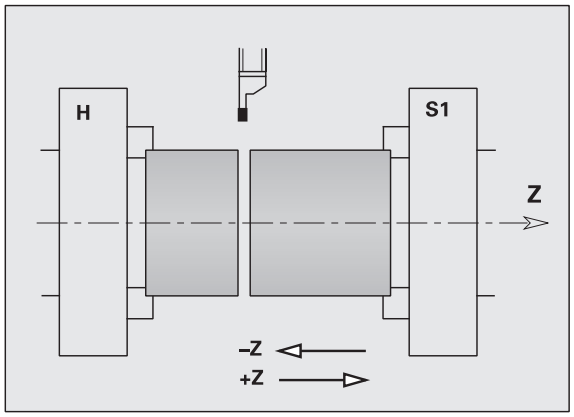


As of software version 625 952-04:

Lag error monitoring is not activated until the acceleration phase has been completed.

Controlled parting using lag error monitoring G917

G917 monitors the path of traverse. The controlled parting function (cut-off control) prevents collisions caused by incomplete parting processes.



Application

- **Parting control** Move the workpiece in the positive Z direction after it has been cut off. If a lag error occurs, the workpiece is defined as **no cut-off.**
- **Smooth parting control:** Move the workpiece in the negative Z direction after it has been cut off. If a lag error occurs, the workpiece is defined as **no correct cut-off.**

In MP 1115, 1165, .. you specify:

- Lag error limit
- Feed rate of the monitored path of traverse

Process of controlled parting:

1. Part the workpiece
2. Activate the monitoring function for the path of traverse with G917
3. Move the workpiece with G1 after it has been cut off
4. The CNC PILOT checks the lag error and stores the result in variable V300
5. Evaluate variable V300

Test results

G917 produces satisfactory results provided that:

- Rough chuck jaws rotate up to 3000 rpm
- Smooth chuck jaws rotate up to 2000 rpm
- Clamping pressure > 10 bars

Programming:

- Program G917 and G1 in one block
- Program G1 .. as follows:
  - For controlled parting: Path > 0.5 mm (to make results)
  - For checking for smooth parting: Path < width of the parting tool
- Result in variable V300
  - 0: Workpiece was not correctly/smoothly cut off (servo lag detected)
  - 1: Workpiece was correctly/smoothly cut off (no servo lag detected)
- G917 generates an interpreter stop

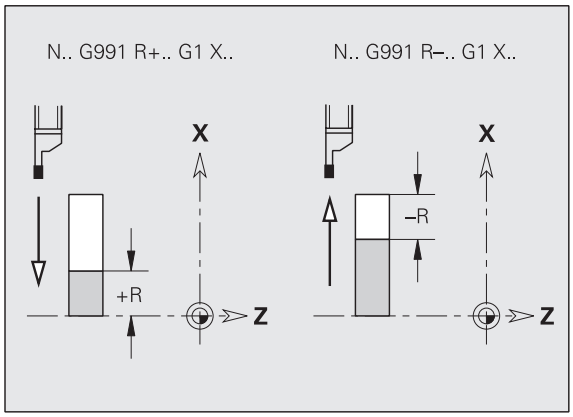


As of software version 625 952-04:

Lag error monitoring is not activated until the acceleration phase has been completed.

Controlled parting using spindle monitoring G991

G991 controls the parting process by monitoring the speed difference between the two spindles. The spindles are connected in terms of actuation via the workpiece. After the workpiece has been cut off, the spindles rotate independently of each other. Speed differences and monitoring time are stored in machine parameters 808, 858, ... They can be changed with G992.



Parameters

R Return path (radius value)

- No input: The difference in speed between the synchronously running spindles is checked (once).
- R>0: Monitoring of the remaining parting path
- R<0: Monitoring of the return path The monitoring begins with the start of the return path and ends at return path – R.

In R you specify the path to be monitored and define whether the path before the workpiece is cut off or the return path is to be monitored (see illustration).

The CNC PILOT writes the monitoring result into variable V300. G991 generates an interpreter stop.

Programming:

- Program constant surface speed G96
- Program G991 and G1 (path before the parting operation or return path) in one block.
- Result in V300:
  - 0: Not cut off
  - 1: Cut off



- Parting control with G917 is preferable to G991.
- Tool breakage results in speed differences which in turn might affect the monitoring result. It is therefore advisable to monitor the reverse path too.

Values for controlled parting G992

G992 overwrites MP 808, MP 858, ... for controlled parting. The new parameters are effective from the next NC block and are retained until G992 is reprogrammed or the parameters are edited.

Parameters

S	Speed difference (in revolutions per minute)
E	Monitoring time (in ms)

4.29 Contour Follow-Up

Automatic contour follow-up is not possible with program branches or repetitions. In these cases you control the contour follow up with the following commands.

Saving/loading contour follow-up G702

G702 saves the current contour or loads a saved contour.

Program G702 for one slide.

Parameters

Q Save/load contour

- Q=0: Saves the current contour. The contour follow-up is not affected.
- Q=1: Loads the saved contour. The contour follow-up is continued with the loaded contour.

Contour follow-up G703

G703 is used to deactivate the contour follow-up.

Parameters

Q Contour follow-up on/off

- Q=0: Off
- Q=1: On

K default branch G706

During program conversion it is unknown which branch of an IF or SWITCH instruction will be run. The initialization of global information such as contour follow-up, rotational speed, incremental positions, etc. is therefore suspended.

With G706 you define the default branch of an IF or SWITCH instruction. This branch is then used to initialize the global information.

Parameters

Q K branch

- Q=0: No default branch defined
- Q=1: THEN branch as default branch
- Q=2: ELSE branch as default branch
- Q=3: Current branch as default branch

Program:

- G706 Q0, Q1, Q2 before the branch
- G706 Q3 at the beginning of the THEN, ELSE or CASE branches

4.30 In-process and Post-process Measuring

In-process measuring

A touch-trigger probe is a prerequisite.

Application example: With in-process measurement you can monitor tool wear. If you use the **tool life monitoring,** worn-out tools are identified as such and the CNC PILOT replaces it.

Example: In-process measuring

`. . .`	
`N.. T..`	Insert the touch probe
`N.. G910`	Activate in-process measurement
`N.. G0 ..`	Pre-position the touch probe
`N.. G912`	
`N.. G1 ..`	Move the touch probe to the workpiece
`N.. G914 G1 ..`	Retract the touch probe
`. . .`	
`N.. G913`	Deactivate in-process measurement
`. . .`	Evaluate the measured values

Switch on in-process measurement G910

G910 activates the probe and the probe monitoring function.

Programming:

- Position the probe at a sufficient distance before the measuring point.
- Program G910 alone in the NC block. G910 is modal
- Program G1 .. as follows:
  - Target position is at a sufficient distance behind the measuring point
  - Activate a per-minute feed rate (G94)

Actual value capture for in-process measurement G912

With G912, when the probe stylus is deflected, the CNC PILOT stops and writes the position into variables V901 to V920. The remaining path of traverse is deleted. You can use Q to influence the reaction to "touch probe not triggered."

Parameters

Q Error evaluation (default: 0)

- Q=0: "Cycle stop" status; the error is displayed
- Q=1: "Cycle on" status; the error number 5518 is saved in variable V982



- X values are measured as a radius.
- The variables are also used by the G functions (G901, G902, G903 and G916). Make sure that your measuring results are not overwritten.

The measuring results are evaluated by the NC program.

If the in-process measurement detects a worn tool, the tool diagnostics sets bit 4 (see "Tool programming" on page 121).

Switch off in-process measuring G913

G913 is used to deactivate probe monitoring. The probe must be reflected before G913 becomes effective. Program only G913 in the NC block. The function generates an interpreter stop.

Switch off probe monitoring G914

After the deflection of the probe stylus, deactivate probe monitoring in order to retract the probe.

Program G914 and G1 in one NC block.

Post-process measurement G915

Post-process measuring means that the workpiece is measured outside the lathe and that the measurement results are transferred to the CNC PILOT.

Prerequisites:

- The measuring device and the CNC PILOT are connected by serial interface
- Communications protocol: 3964-R

The measuring device determines whether measured values or compensation values are transferred. The measurement results are evaluated by the NC program. A **global measurement result** should be transferred to measuring position 0.

Parameters

H Block

- H=0: Reserved
- H=1: Pending measured values are uploaded

With G915, measured values are received from the post-process measuring system and stored in the following variables:

- V939: Global measurement result
- V940 Measuring status
  - 0: **No** new measured values
  - 1: New measured values
- V941 to V956 (corresponding to measuring points 1 to 16).

You can use the **tool life monitoring** in connection with post-process measurement. If a tool is identified as worn out, the CNC PILOT inserts a replacement tool.

If the post-process measurement detects a worn tool, the tool diagnostics sets bit 5 (see “Tool programming” on page 121).



- The post-process measuring status as well as the measured values last received can be checked in Automatic mode.
- Evaluate the measurement status in order to avoid that compensation values are accounted for twice or incorrectly.

Example: Using a measuring result as compensation value

. . .	
N2 T1	Finish the outside contour
. . .	
N49 ...	End the workpiece machining
N50 G915 H1	Request the measurement results...
N51 IF {V940==1}	...if there are any results
N52 THEN	
N53 V {D1 [X] = D1 [X] + V941}	Add result to compensation value D1
N54 ENDIF	
. . .	

Example: Monitoring for tool breakage

. . .	
N2 T1	Rough the outside contour
. . .	
N49 ...	End the workpiece machining
N50 G915 H1	Request the measurement results...
N51 IF {V940==1}	...if there are any results
N52 THEN	
N53 V {V941 >= 1}	Measured value > 1mm
N54 THEN	
N55 PRINTA	“Measured value > 1 mm = tool breakage”
N56 M0	Programmed stop—cycle off
N57 ENDIF	
N58 ENDIF	
. . .	

4.31 Load Monitoring

Fundamentals of load monitoring

The load monitoring function checks the performance and work values of the drives and compares them to limit values which have been determined during a **reference machining cycle.**

The CNC PILOT considers two limit values:

- If the first limit value is exceeded, the tool is marked as worn out and the **tool life monitoring** inserts the replacement tool (see "Tool programming" on page 121).
- Second limit value exceeded: The load monitor reports a broken tool and stops the program run (cycle stop).

Example of load monitoring

`. . .`	
`N.. G996 Q1 H1`	Torque monitoring, excluding rapid traverse paths
`. . .`	
`N.. G14 Q0`	
`N.. G26 S4000`	
`N.. T2`	
`N.. G995 H1 Q9`	Monitor the main spindle and X axis
`N.. G96 S230 G95 F0.35 M4`	
`N.. M108`	
`N.. G0 X106 Z4`	
`N.. G47 P3`	
`N.. G820 NS..`	Monitor the feed paths of the roughing cycle
`N.. G0 Z4`	
`N.. M109`	
`N.. G995`	End of monitoring zone
`. . .`	

Specifying the monitoring zone G995

G995 defines the monitoring zone and the axes to be monitored.

- G995 with parameters: Start of monitoring zone
- G995 without parameters: End of the monitoring zone (not necessary if another monitoring zone follows

Parameters

H Number of the monitoring zone (1 <= H <= 999)

Q Code for axes (drives to be monitored)

- 1: X axis
- 2: X axis
- 4: Z axis
- 8: Main spindle
- 16: Spindle 1
- 128: C axis 1

The zone number must be unambiguous in the NC program. A maximum of 49 monitoring zones per slide are possible.



- Add the codes if you want to monitor more than one drive. (Example: Monitoring the Z axis and main spindle: Q=12)
- The code for axes is defined in bit numbers for load monitoring (control parameter 15).

Type of monitoring G996

G996 defines the type of monitoring or switches the load monitoring off temporarily.

Parameters

Q Scope of monitoring (default: 0)

- Q=0: Monitoring not active (applies for the entire NC program; even previously programmed G995 commands are without effect)
- Q=1: Do not monitor rapid traverse
- Q=2: Monitor rapid traverse

H Type of monitoring (default: 0)

- H=0: Torque monitoring and work monitoring
- H=1: Torque monitoring
- H=2: Work monitoring

4.32 Other G Functions

Period of dwell G4

With G4, the CNC PILOT interrupts the program run for the time F before executing the next program block. If G4 is programmed together with a path of traverse in the same block, the dwell time only becomes effective after the path of traverse has been executed.

Parameters

F Dwell time [sec] (0 < F <= 999)

Precision stop G7

G7 switches precision stop on. It is a modal function. With a precision stop the CNC PILOT does not run the following block until the last point has been reached in the tolerance window for position (tolerance window: MP 1106, .. "position control for linear axis").

Precision stop affects single contours and cycles. The NC block containing G7 is also executed with a precision stop.

Precision stop off G8

G8 switches precision stop off. The block containing G8 is executed **without** a precision stop.

Precision stop G9

G9 activates a precision stop for the block in which it is programmed. With a precision stop the CNC PILOT does not run the following block until the last point has been reached in the tolerance window for position (tolerance window: MP 1106, .. "position control for linear axis").

Move rotary axis G15

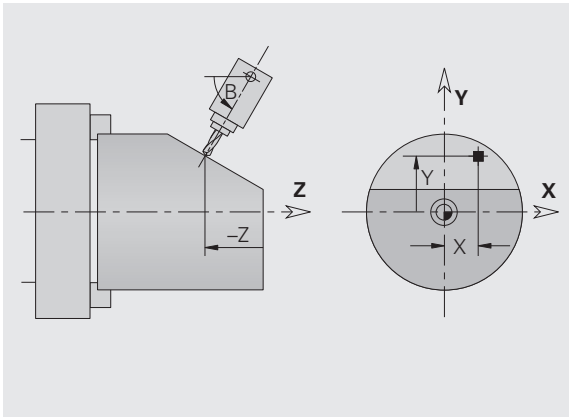
G15 tilts the rotary axis to the given angle and moves at feed rate to the programmed position.

Parameters

A, B	Angle – end point of rotary axis
X, Y, Z	End point of principal axis (X diameter value)
U, V, W	End point of secondary axis



Use G15 to move to position, not to machine



Switch off protection zone G60

G60 is used to cancel protection zone monitoring. G60 is programmed **before** the traversing command to be monitored or not monitored.

Parameters

Q Activate/Deactivate

- Q=0: Activate protection zone (modal)
- Q=1: Deactivate protection zone (modal)

Application example: With G60, you can temporarily deactivate a programmed monitoring of the protection zone in order to machine a centric through hole.

Example: G60

```
. . .
N1 T4 G97 S1000 G95 F0.3 M3
N2 G0 X0 Z5
N3 G60 Q1  [Deactivate the protection zone]
N4 G71 Z-60 K65
N5 G60 Q0    [Activate the protection zone]
. . .
```

Chucking equipment in simulation G65

G65 displays the selected chucking equipment in the simulation graphics. G65 needs to be programmed separately for each chuck. G65 H.. without X, Z cancels the chuck in the simulation graphics.

Parameters

H	Chuck number (H=1 to 3; reference to CHUCKING EQUIPMENT)
X	Starting point—Chuck reference point (diameter value)
Z	Starting point—Chuck reference point
D	Spindle number (reference: CHUCKING EQUIPMENT section)
Q	Chuck shape (only for chuck jaws)—(default: Q from the CHUCKING EQUIPMENT section).

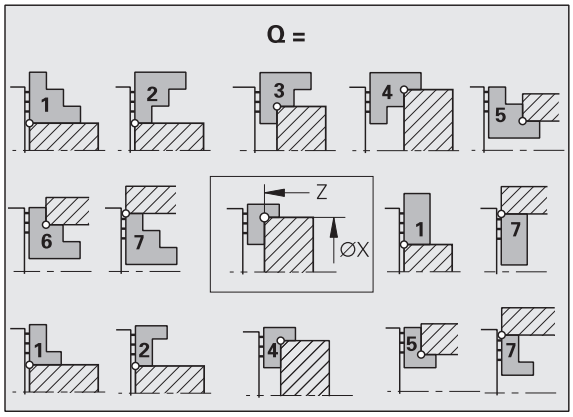
Chucks are described in the database and are defined in the CHUCKING EQUIPMENT section (H=1 to 3).

The **Chuck reference point** determines the position of the chuck in the simulation graphics. The position of the reference point depends on the chuck shape (see illustration). The chuck reference point is dimensioned with respect to the workpiece zero point.

The CNC PILOT mirrors the chucks H=1 to 3 if they are positioned to the right of the workpiece.

Note on the simulation graphics and the reference point:

- H=1 – Chuck:
  - Is depicted open
  - Reference point X: Center of chuck
  - Reference point Z: “Right edge” (take into account the width of the chuck jaws)
- H=2 – Chuck jaw (Q defines the reference point and inside/outside clamping):
  - Position of the reference point: See illustration for G65
  - Inside clamping: 1, 5, 6, 7
  - Outside clamping: 2, 3, 4
- H=3 – Chucking accessory (dead center, lathe center, etc.):
  - Reference point in X: Center of the chucking equipment
  - Reference point in Z: Point of the chucking equipment



Example: G65

```
. . .
SPANNMITTEL 1 [CHUCKING EQUIPMENT]
H1 ID"KH110"                    [Chuck]
H2 ID"KBA250-77"             [Chuck jaw]
H4 ID"KSP-601N"             [Dead center]
. . .
ROHTEIL [WORKPIECE BLANK]
N.. G20 X80 Z200 K0
. . .
BEARBEITUNG [MACHINING]
$1 N.. G65 H1 X0 Z-234
$1 N.. G65 H2 X80 Z-200 Q4
. . .
```

On lathes with multiple slides, program the NC blocks containing G65 with “slide code $..”. Otherwise, the system displays more than one chuck.

Component position G66

The simulation can only depict tool positions and movements if the X and Z position, or the X, Y and Z position, is known. For slides that move in only one direction (for example a parting slide), add the missing coordinates with G66. In "Shift" you can include a zero point shift. On the basis of these data, the CNC PILOT simulates slides with one axis.

Parameters

X	Starting point Component position G66
I	Shift
Z	Starting point Component position G66
K	Shift
Y	Starting point Component position G66
J	Shift

Waiting for time G204

G204 interrupts an NC program up to a specified moment.

Parameters

D	Tag [1-31] (default: First possible time H, Q)
H	Hour [0-23]
Q	Minute [0-59]

Update nominal values G717

G717 is used to update the nominal position values of the control with the axis position data.

Application:

- Deleting the lag error.
- Standardization of the slave axes after deactivating a master-slave coupling.



Use G717 only in "expert programs."

Move lag error G718

G718 prevents the automatic updating of nominal control position values with the axis position data (e.g. when traversing to a fixed stop, or after canceling and re-enabling a controller release).

Parameters

Q On / Off

- Q=0: Off
- Q=1 on, the servo lag stays saved

Application:
Before activating a master-slave axis coupling.



Use G718 only in “expert programs.”

Actual values in variables G901

G901 is used to transfer the actual values to the variables V901 to V920.

The function generates an interpreter stop.

Zero-point shift in variables G902

G902 writes the shift in **Z direction** into variables V901 to V920.

The function generates an interpreter stop.

Lag error in variables G903

G903 transfers the current following error (distance by which the actual values lags the nominal value) into the variables V901..V920.

The function generates an interpreter stop.

Block speed monitoring off G907

The CNC PILOT starts traverse paths requiring spindle revolutions when the programmed speed has been reached. G907 is used to deactivate speed monitoring block by block. The path of traverse is started immediately.

Program G907 and the traverse path in the same NC block.

Feed rate override 100% G908

G908 sets the feed override for traverse paths (G0, G1, G2, G3, G12, G13) block by block to 100%.

Program G908 and the traverse path in the same NC block.

Interpreter stop G909

The CNC PILOT pre-interprets approx. 15 to 20 NC blocks. If variables are assigned shortly before the evaluation, "old values" would be processed. G909 stops the pre-interpretation. The NC blocks are processed up to G909. Only after G909, are the subsequent NC blocks processed.

Apart from G909, the NC block should only contain synchronous functions. (Some G functions generate an interpreter stop.)

Velocity feedforward G918

G918 is used to deactivate the feedforward. Program G918 in a separate NC block program before/after the thread cutting (G31, G33).

Parameters

Q Feedforward off/on (default: 1)

- Q=0 off
- Q=1 on

Spindle override 100% G919

G919 is used to deactivate the spindle speed override.

Parameters

Q Spindle number (default: 0)

H Type of limit (default: 0)

- H=0: Switch-on spindle override
- H=1: Spindle override at 100% – modal
- H=2: Spindle override at 100% – for the current NC block

Deactivate zero-point shifts G920

G920 deactivates the workpiece zero point and zero-point shifts. Traverse paths and position values are referenced to the **distance tool tip – machine zero point.**

Deactivate zero-point shifts, tool lengths G921

G921 deactivates the workpiece zero point, zero point shifts and tool dimensions. Traverse paths and position values are referenced to the **slide reference point – machine zero point.**

T no. internal G940

G940 finds the actual magazine tool to be inserted. G940 is usually used in the context of expert programs with plate magazines.

Parameters

P Tool number of the form of “mmDDpp”

- mm: Pocket number on the plate magazine
- DD: Position in the magazine list
- pp: Turret pocket. For a tool holder, pp=01

If the tool life management is used, a replacement tool is inserted as soon as the programmed tool age has expired. G940 finds the actual tool to be inserted. The programmed tool is transferred in P. In reply, the tool actually to be inserted is written in the following variables:

- V311: pp
- V312: dd
- V313: mm
- V331: mmddpp

Transferring magazine compensation values G941

G941 writes in the following variables the compensation values of the magazine tool to be deposited and to be fetched. The compensation values describe the deviations of the individual magazine pockets from the standard dimensions.

In V800, write the number of the tool to be deposited, and use G940 to find the tool to be fetched before you program G941.

- Compensation values of tool to be fetched:
  - V931: Compensation in X
  - V932: Compensation in Z
  - V933: Compensation in Y
  - V934: Compensation in C
- Compensation values of tool to be deposited:
  - V935: Compensation in X
  - V936: Compensation in Z
  - V937: Compensation in Y
  - V938: Compensation in C

Servo lag limit G975

G975 switches over to servo-lag limit 2 (MP 1106, ..). G975 is a modal function. At the end of a program the CNC PILOT switches to the standard lag error limit.

Parameters

H Servo lag limit (default: 1)

- H=1 Standard servo lag limit
- H=2 Following error limit 2

Activating zero-point shifts G980

G980 activates the workpiece zero point and all zero offsets. Traverse paths and position values are referenced to the distance of the **tool tip to the workpiece zero point,** while taking the zero point shifts into consideration.

Activate zero-point shifts, tool lengths G981

G981 activates the workpiece zero point, all zero-point shifts and the tool dimensions. Traverse paths and position values are referenced to the distance of the **tool tip to the workpiece zero point,** while taking the zero point shifts into consideration.

Sleeve monitoring G930

G930 activates/deactivates the sleeve monitoring. When the monitoring is activated, the maximum contact force for one axis is defined. The sleeve monitoring can be activated for only **one** axis per NC channel.

Parameters

X/Y/Z Contact force [dN] – The contact force is limited to the given value

- 0: Deactivate the sleeve monitoring
- >0: Contact force is monitored

Application example: G930 is applied to use the opposing spindle as a mechatronic tailstock. In this case the opposing spindle is equipped with a dead center and the contact force is limited with G930. A prerequisite for this application is a PLC program from the machine tool builder that enables the user to operate the mechatronic tailstock in the Manual and Automatic operating mode.



As of software version 625 952-04:

Lag error monitoring is not activated until the acceleration phase has been completed.

Shaft speed with V constant G922

Available as of software version 625 952-05.

With constant surface speed (V constant), the spindle speed depends on the X position of the tool tip. With G922 you program whether this procedure should apply to G0 paths, as well.

G922 applies to the spindle assigned to the slide.

Parameters

H Method of optimization

- 0: Standard behavior
- 1: Optimized spindle speed with G0 paths
- 2: Speed optimization with G0 paths (V constant)

Optimized spindle speed: At the transition from traverse path to rapid travers path, the spindle speed is frozen at the speed of the last traverse path. This spindle speed is maintained for further rapid traverse paths. Only during the last rapid traverse path before the transition to the traverse path (new approach) will the spindle speed depend again on the X postion of the tool tip.

Speed optimization with G0 paths: The spindle speed depends on the X position of the tool tip.

G922 is permanently effective. It remains effective until the next G922 or program end.

If G922 is **not** used, the following "standard behavior" is applicable:

- Machines with **one slide**: For G0 paths, the principle of the optimized spindle speed is applied:
- Machines with **several slides**, also including **several slides with X axis**: V constant is also applicable to G0 paths.
- Machines wiht **several slides** but only **one slide with X axis**: The behavior is dependent on machine parameter 18, Bit 8.
  - Bit 8=0: V constant is also applicable to G0 paths.
  - Bit 8=1: For G0 paths, the principle of the optimized spindle speed is applied:

4.33 Data Input and Data Output

Output window for # variables WINDOW

WINDOW (x) opens an output window with x lines. The window is opened as a result of the first input/output. WINDOW (0) closes the window.

Syntax:

WINDOW(line number) (0 <= line number <= 10)

The standard window comprises 3 lines. You do not need to program it.

Example:

```
. . .
N.. WINDOWS(8)
. . .
N.. INPUT(input diameter: ,#1)
. . .
N.. PRINT(output diameter: ,#1)
. . .
```

Input of # variables INPUT

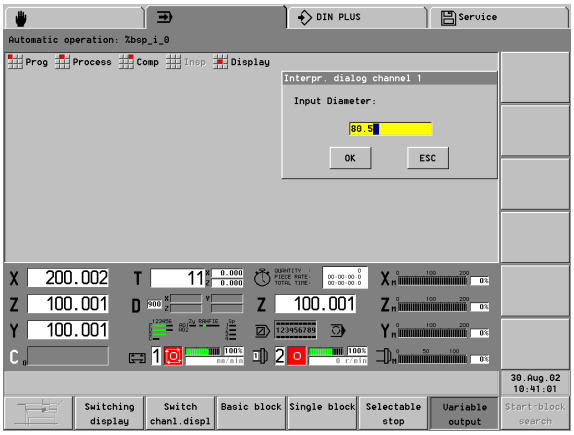
With INPUT you program the input of # variables that are evaluated during program interpretation.

Syntax:

INPUT(Text,variable)

You define the input text and the number of the variable. The CNC PILOT stops the interpretation at INPUT, outputs the text and waits for input of the variable value.

The CNC PILOT displays the input after having completed the INPUT command.

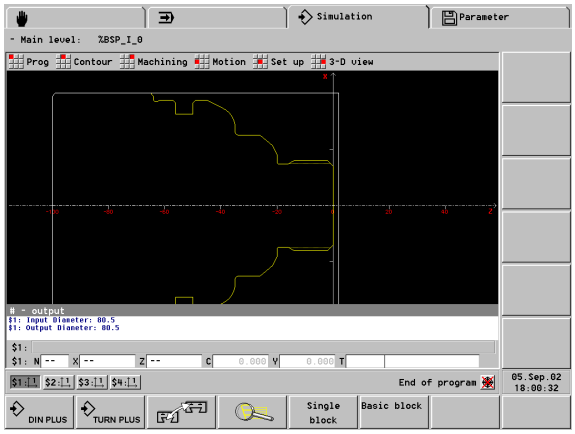


Output of # variables PRINT

PRINT can be used to output texts and variable values during program interpretation. You can program a succession of several texts and # variables.

Syntax:

PRINT(text,variable, text,variable, ..)



V variable simulation

V variables and all data input/output are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.

Output window for V variables WINDOWA

WINDOWA (x) opens an output window with x lines. The window is opened as a result of the first input/output. WINDOWA (0) closes the window.

Syntax:

WINDOWA(line number) – (0 <= line number <= 10)

The standard window comprises 3 lines. You do not need to program it.

Example:

```
. . .
N.. WINDOWSA(8)
. . .
N.. INPUTA(input diameter: ,#1)
. . .
N.. PRINTA(output diameter: ,#1)
. . .
```

Input of V variables INPUTA

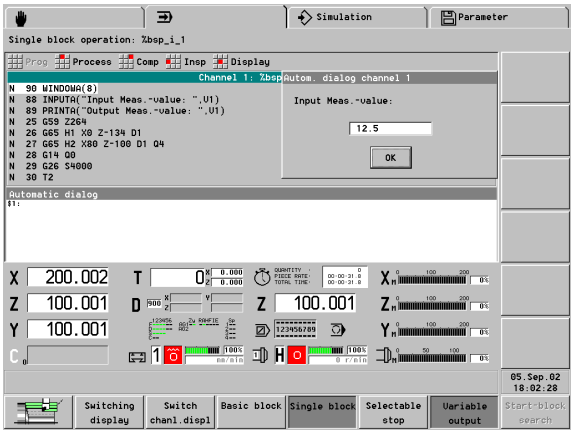
With INPUTA you program the input of V variables that are evaluated during program interpretation.

Syntax:

INPUTA(Text,variable)

You define the input text and the number of the variable. The CNC PILOT requests the input of the variable value during the execution of the command. The input is assigned to the variable and the program run continues.

The CNC PILOT displays the input after having completed the INPUT command.



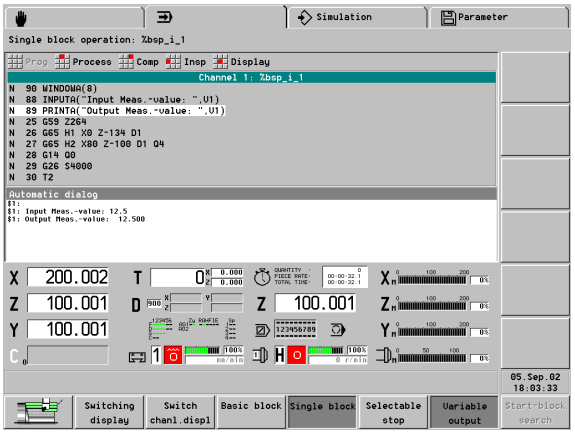
Output of V variables PRINTA

PRINTA can be used to display text and V-variable values in an output window. You can program a succession of several texts and variables.

Syntax:

PRINTA(text,variable, text,variable, ..)

To output texts and variable values on a printer, activate printer output in control parameter 1 (printer output ON).



4.34 Programming with Variables

The CNC PILOT supports NC programs before the program run. The system therefore differentiates between two types of variables:

- # variables are evaluated during **NC program interpretation.**
- V variables (or events) are evaluated during **NC program run.**

These rules apply during calculation:

- Multiplication/division before addition/subtraction
- Up to 6 bracket levels
- **Integer variables** (only for V variables): Integer values between –32767 .. +32768
- **Real variables:** Floating point numbers with max. 10 integers and 7 decimal places
- The variables are retained even if the control has been switched off in the meantime.
- See the table for the available mathematical operations



Program NC blocks containing variable calculations with slide code $.. if your lathe has more than one slide. Otherwise, the calculations are repeated.

Syntax	Mathematical functions
+	Addition
–	Subtraction
*	Multiplication
/	Division
SQRT(...)	Square root
ABS(...)	Absolute amount
TAN(...)	Tangent (in degrees)
ATAN(...)	Arc tangent (in degrees)
SIN(...)	Sine (in degrees)
ASIN(...)	Arc sine (in degrees)
COS(...)	Cosine (in degrees)
ACOS(...)	Arc cosine (in degrees)
ROUND(...)	Round
LOGN(...)	Natural logarithm
EXP(...)	Exponential function ex
INT(...)	Truncate decimal places
Only with **# variables:**	
SQRTA(.., ..)	Square root of (a^2+b^2)
SQRTS(.., ..)	Square root of $(a^2–b^2)$

variables

The CNC PILOT uses value ranges to define the **scope of variables:**

- **#0 to #29 Channel-dependent, global variables** can be used for each slide (NC channel). Identical variable numbers on different slides are no problem. The variables are retained after the program has been completed and can be processed by the following NC program.
- **#30 to #45 Channel-independent, global variables** are available once within the control. If the NC program changes a variable, it applies to all slides. The variables are retained after the program has been completed and can be processed by the following NC program.
- **#46 to #50 Variables reserved for expert programs:** Do not use these variables in your NC program.
- **#256 to #285 Local variables** are effective only within a subprogram.



Positions and dimensions are always indicated in metric form. This also applies when an NC program is run in inches.

NC information in # variables	
#768, #770	Last programmed position X (radius value), Y, Z
#771	Last programmed position C [°]
#772	Active operating mode: ■ 2: Machine ■ 3: Simulation ■ 4: TURN PLUS
#774	TRC/MCRC status: ■ 40: G40 active ■ 41: G41 active ■ 42: G42 active
#775	Number of the selected C axis
#776	Active wear compensation (G148): ■ 0: DX, DZ ■ 1: DX, DZ ■ 2: DX, DS
#778	Unit of measure: 0=metric; 1=inch
#782	Active working plane: ■ 17: XY plane (front or back) ■ 18: XZ plane (turning) ■ 19: YZ plane (side view/surface)
#783, #785, #786	Distance between tool tip and slide zero point Y, Z, X

Read parameter values in # variables

Syntax: **#1 = PARA(x,y,z)**

- x = Parameter group
  - 1: Machine parameters
  - 2: Control parameters
  - 3: Setup parameters
  - 4: Machining parameters
  - 5: PLC parameters
- y = Parameter number
- z = Sub-parameter number

As of software version 625 952-02:
Check numerical value for specified bit

Syntax: **#1 = BITSET(x,y)**

- x = Bit number (0 to 15)—can be replaced by a # variable.
- y = Numerical value (0 to 65535)—can be replaced by a # variable.

The function returns 1 if the numerical value contains the interrogated bit. Otherwise, 0 is returned.

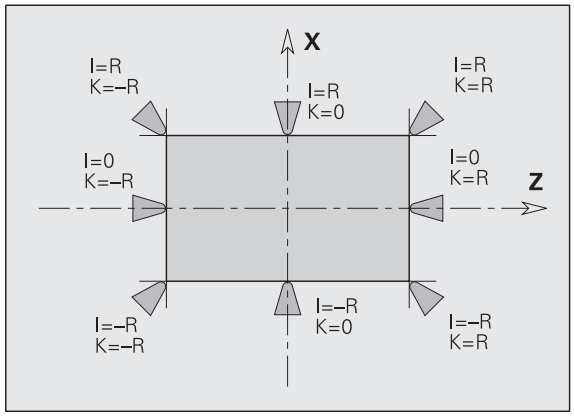
Bit => Numerical value:	
0 => 1	1 => 2
2 => 4	3 => 8
4 => 16	5 => 32
6 => 64	7 => 128
8 => 256	9 => 512
10 => 1024	11 => 2048
12 => 4096	13 => 8192
14 => 16384	15 => 32768

Example:

```
. . .
[reads machine dimension 1 Z  in variable
#1]
N.. #1=PARA(1,7,2)
. . .
N.. #1=#1+1
N.. G1 X#1
N.. G1 X(SQRT(3*(SIN(30)))
N.. #1=(ABS(#2+0.5))
. . .
```

NC information in # variables	
#787	Reference diameter for lateral surface machining (G120)
#788	Spindle holding the workpiece (G98)
#790	Oversize G52-Geo ■ 0: Do not include ■ 1: Include
#791..#792	G57 oversizes X, Z
#793	G58 oversize P
#794..#795	Cutting width in X and Z by which the tool reference point is shifted with G150/G151
#796	Number of spindle for which the last feed rate was programmed
#797	Number of spindle for which the last speed was programmed
#801	Tilted plane is active
#802	■ 0: G30 not active ■ 1: G30 active
#803	Number of the selected language, based on the sequence of languages specified in control parameter 4 (starting with 0)
#804	Is DataPilot? ■ 0: Control ■ 1: DataPilot

Tool information in # variables	
#512	Tool type (3-digit number)
#513..#515	1., 2., 3. position of tool type
#516	Usable length (nl) for turning, drilling or boring tools:
#517	Main working direction: ■ 0: Undefined ■ 1: +Z ■ 2: +X ■ 3: –Z ■ 4: –X ■ 5: +/–Z ■ 6: +/–X
#518	Secondary machining direction of turning tools
#519	Depending on the tool type: ■ 14*: 1 = right, 2 = left execution (A) ■ 5**, 6**: Number of teeth



Tool information in # variables	
#520	Depending on the tool type: ■ 1**, 2**: Tool tip radius (rs) ■ 3**, 4**: Shank diameter (d1) ■ 51*, 52*: Cutter diameter in front (df) ■ 56*, 6**: Cutter diameter (d1)
#521	Depending on the tool type: ■ 11*, 12*: Shaft diameter (sd) ■ 14*, 15*, 16*, 2**: Cutting width (sb) ■ 3**, 4**: Length of first cut (al) ■ 5**, 6**: Milling width (fb)
#522	Tool orientation (reference: machining direction of tool): ■ 0: On the contour ■ 1: To the right of the contour ■ – 1: To the left of the contour
#523..#524	Set-up dimensions (ze, xe, ye)
#526..#527	Position of tool tip center I, K (see illustration)
#780	Tool from the database

Precondition for tool information: A tool call must be programmed for the variables to become effective.

V variables

The CNC PILOT uses value ranges to define the **scope of variables:**

- Real: V1 to V199
- Integer: V200 to V299
- Reserved: V300 to V900

The PLC program reads and describes the variables V1..V299.

Requests and assignments	
Read/write machine dimensions (MP 7):	
Syntax:	**V{Mx[y]}** ■ x = Dimension 1 to 9 (10 to 99 only for machine tool builders) ■ y = coordinate: X, Y, Z, U, V, W, A, B or C
Read/write tool compensation:	
Syntax:	**V{Dx[y]}** ■ x = T number ■ y = length compensation: X, Y, or Z

Requests and assignments

Interrogating sequential events:

Syntax: **V{Ex[1]}**

- x = event: 20 to 59, 90
  - 20: Life of this tool has expired (global information)
  - 21 to 59: Tool life **of this** tool has expired
  - 90: Start block search (0=not active; 1=active)

Interrogate external events:

Syntax: **V{Ex[y]}**

- x = slide 1 to 6
- y = bit: 1 to 16

Interrogates **bit** of the event for 0 or 1. The meaning of the external event is determined by the machine manufacturer.

Read from/write to tool diagnosis bits:

Syntax: **V{Tx[y]}**

- x = T number
- y = bit: 1 to 16 (bit=0: no; bit=: yes)
  - Bit 1: Tool worn out (reason for replacing a tool: see bits 2 to 8)
  - Bit 2: Specified tool life/piece number has been reached
  - Bit 3: Tool wear determined by in-process measuring of tool.
  - Bit 4: Tool wear determined by in-process measuring of workpiece.
  - Bit 5: Tool wear determined by post-process measuring of workpiece.
  - Bit 6: Tool breakage identified by the load monitoring function.
  - Bit 7: Tool wear identified by the load monitoring function.
  - Bit 8: A neighboring edge of the multipoint tool is worn-out
  - Bit 9: Cutting edge new?
  - Bit 12: The remaining tool life of the cutting edge is <6% or the remaining piece number is 1

Bits 9 to 16 contain general information.



- Take into account the advance interpretation of the NC blocks when working with V variables and, if necessary, program an interpreter stop (see "Interpreter stop G909" on page 307).
- The content of the variables is retained even if the control has been switched off. If required, initialize the variables at the beginning of the program in order to prevent undefined variable contents.

Sequential events and tool life monitoring

- The tool life monitoring function and the function for searching the start block trigger sequential events.
- Assign this clock event to the tool ("tool life management"—Manual control mode).
- When a tool is worn-out, event 20 (global information) and event 1 are triggered. Event 1 identifies the worn-out tool. When the last tool of an tool interchange chain is worn out, event 2 is also triggered.
- Define events 1 and 2 for each individual tool of the tool interchange chain.
- The CNC PILOT resets the sequential events at the program end (M99).



- If an tool interchange chain is defined, program the first tool of the chain for tool compensation and tool diagnosis. The CNC PILOT addresses the **active tool of the tool interchange chain** (see "Tool programming" on page 121).
- **Machine dimensions:** Please note the reference points. Example: You teach-in a position value with respect to the machine zero point. You should then approach this machine dimension also with respect to the machine zero point.

The CNC PILOT saves the various information in variables that you can read into the NC program (see table).

Information in V variables	
V660	Unit count: ■ Is set to 0 during system start and when loading a new NC program. ■ Is increased by 1 with M30, M99 and a counting pulse (M18). The piece counter in V660 is different from the piece counter in the machine display.
V840.. V843	G901, G902 and G903 write the positions of the secondary axes of the calling channel into the variables: ■ Auxiliary axis 1 ■ Auxiliary axis 2 ■ Auxiliary axis 3 ■ Auxiliary axis 4

Information in V variables	
V901 to V920	G901, G902, G903, G912 and G916 write the positions into the variables: ■ V901 to V903: Axis X, Z, Y of slide 1 ■ V904 to V906: Axis X, Z, Y of slide 2 ■ V907 to V909: Axis X, Z, Y of slide 3 ■ V910 to V912: Axis X, Z, Y of slide 4 ■ V913 to V915: Axis X, Z, Y of slide 5 ■ V916 to V918: Axis X, Z, Y of slide 6 ■ V919: C axis 1 ■ V920: C axis 2 X values are saved as radius values. The variables are overwritten, even is they have not yet been evaluated.
V921	Measuring angular offset during G906, spindle synchronization
V922/ V923	Event with G905 C-angle offset
V982	Error number with G912, actual value determination for in-process measurement
V300	Result of G991, controlled parting

Example for V variables

`. . .`	
`N.. V{M1[Z]=300}`	Sets machine dimension 1 Z to 300
`N.. G0 Z{M1[Z]}`	Moves to machine dimension 1 Z
`N.. IF{E1[1]==0}`	Interrogation of external event 1 – bit 1
`N.. V{D5[X]=1.3}`	Sets compensation X for tool 5
`N.. V{V12=17.4}`	
`N.. V{V12=V12+1}`	
`N.. G1 X{V12}`	
`. . .`	

4.35 Conditional Block Run

Program branching IF..THEN..ELSE..ENDIF

A conditional branch consists of the elements:

- IF, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- THEN. If the condition is fulfilled, the THEN branch is executed
- ELSE. If the condition is not fulfilled, the ELSE branch is executed
- ENDIF concludes the conditional program branch.

Programming:

- Select “Instr. > DIN PLUS words” in the machining menu. The CNC PILOT opens the “DIN PLUS words” selection list.
- Select IF
- Enter the condition
- Insert NC blocks of the THEN branch.
- If required: Insert NC blocks of the ELSE branch.

V variables are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.



- NC blocks with IF, THEN, ELSE, ENDIF can have no further commands.
- You can combine up to two conditions.
- For branches due to V variables or events, the contour follow-up is switched off with the IF statement and back on with the ENDIF statement. With G702, G703 or G706 you control the contour follow-up.

Relational operators for	
<	Less than
<=	Less or equal
<>	Not equal
>	Greater than
>=	Greater than or equal
==	Equal
Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

Example: IF..THEN..ELSE..ENDIF

```
. . .
N.. IF{E1[16]==1}
N.. THEN
N..     G0 X100 Z100
N.. ELSE
N..     G0 X0 Z0
N.. ENDIF
. . .
```

WHILE..ENDWHILE program repeat

A program repeat consists of the elements:

- WHILE, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- ENDWHILE concludes the conditional program repeat.

NC blocks programmed between WHILE and ENDWHILE are executed repeatedly for as long as the condition is fulfilled. If the condition is not fulfilled, the CNC PILOT continues execution of the program with the block programmed after ENDWHILE.

Programming:

- Select “Instr. > DIN PLUS words” in the machining menu. The CNC PILOT opens the “DIN PLUS words” selection list.
- Select WHILE
- Enter the condition
- Insert NC blocks between WHILE and ENDWHILE.

V variables are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.



- You can combine up to two conditions.
- If the repetition is made due to V variables or events, the contour follow-up function is switched off for the WHILE branch and switched on again with ENDWHILE. With G702, G703 or G706 you control the contour follow-up.
- If the condition you program in the WHILE command is always true, the program remains in an endless loop. This is one of the most frequent causes of error when working with program repeats.

Relational operators	
<	Less than
<=	Less or equal
<>	Not equal to
>	Greater than
>=	Greater than or equal
==	Equal
Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

Example: WHILE..ENDWHILE

```
. . .
N.. WHILE (#4<10) AND (#5>=0)
N..     G0 Xi10
. . .
N.. ENDWHILE
. . .
```

SWITCH..CASE—program branching

The switch statement consists of the elements:

- SWITCH, followed by a variable. The content of the variables is interrogated in the following CASE instruction.
- CASE x: The CASE branch is run with the variable value x. CASE can be programmed repeated times.
- DEFAULT: This branch is executed if no CASE instruction matched the variable value. DEFAULT can be omitted.
- BREAK: Concludes the CASE branch or DEFAULT branch

Programming:

- Select "Instr. > DIN PLUS words" in the machining menu. The CNC PILOT opens the "DIN PLUS words" selection list.
- Select SWITCH
- Enter the switch variable
- For each CASE branch:
  - Select CASE (from the DIN PLUS words selection list)
  - Enter the SWITCH condition (value of the variable) and insert the NC blocks to be executed.
- For the DEFAULT branch: Insert the NC blocks to be executed.

V variables are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.



- You can combine up to two conditions.
- If the branch is triggered by V variables or events, the contour follow-up is switched off with the SWITCH statement and back on with the ENDSWITCH statement. With G702, G703 or G706 you control the contour follow-up.

Relational operators	
<	Less than
<=	Less or equal
<>	Not equal to
>	Greater than
>=	Greater than or equal
==	Equal
Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

Example for V variables

```
. . .
N.. SWITCH{V1}
N..   CASE 1    [EXECUTED IF V1=1]      Executed if V1=1
N..     GO XI10
. . .
N..   BREAK
N..   CASE 2    [EXECUTED IF V1=2]      Executed if V1=2
N..     GO XI20
. . .
N..   BREAK
N..   DEFAULT                           No CASE instruction matched the variable value
N..     GO XI30
. . .
```

```
N..   BREAK
N..   ENDSWITCH
. . .
N..   DEFAULT
N..     G0 XI30
. . .
N..   BREAK
N..   ENDSWITCH
. . .
```

Skip level /..

An NC block with prepended skip level is **not executed** if the skip level **is active.** Activate/deactivate the skip levels in Automatic mode.

In addition, you can use the **skip cycle (deletion cycle)** (set-up parameter 11: deletion level/cycle). Skip cycle x activates the skip level every xth time.

Example: /1 N 100 G...

N100 .. is **not** executed with active skip cycle 1.

Slide code $..

An NC block preceded by a slide code is executed only for the indicated slide. NC blocks that are not preceded by slide codes are executed on all slides.



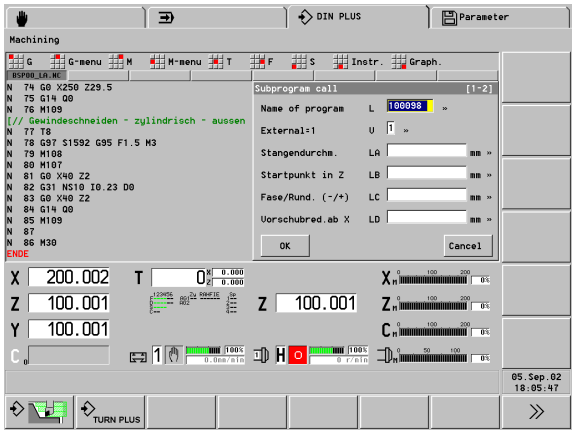
For lathes equipped with one slide or with program heads containing only **one** slide, a slide code is not necessary.

4.36 Subprograms

Subprogram call: L"xx" V1

The subprogram contains the following elements:

- L: Identifying letter for subprogram call
- "xx": Name of the subprogram – file name for external subprograms (max. 8 letters or numbers)
- V1: Identification code for **external** subprograms—omitted for local subprograms



Note on using subprograms:

- External subprograms are defined in a separate file. They are called from any main program, other subprograms, or from TURN PLUS.
- Local subprograms are in the main program file. They can be called only from the main program.
- Subprograms can be nested up to 6 times. Nesting means that another subprogram is called from within a subprogram.
- Recursion should be avoided.
- You can add up to 20 transfer values to a subprogram.
  - Designations: LA to LF, LH, I, J, K, O, P, R, S, U, W, X, Y, Z
  - The identification code within the subprogram is: #__.., followed by the parameter designation in lowercase letters (for example: #__la).
  - Use the transfer values when programming with variables within the subprogram.
- The variables #256 – #285 are available in every subprogram as local variables.
- If a subprogram is to be executed repeatedly, define in the number repeats Q parameter the number of times the subprogram is to be repeated.
- A subprogram ends with RETURN.

The parameter LN is reserved for the transfer of block numbers. This parameter may receive a new value when the NC program is renumbered.

Dialog texts in subprogram call

You can define up to 19 parameter descriptions that precede/follow the input fields in an external subprogram. The CNC PILOT automatically sets the unit of measure for parameter values to the metric system or inches.

The parameter descriptions can be positioned within the subprogram as desired.

Parameter descriptions (see table at right):

[//] – Beginning

[pn=n; s=parameter text (up to 16 characters)]

[//] – End

pn: Parameter designations (la, lb, ...)

n: Conversion number for units of measurement

- 0: Non-dimensional
- 1: mm or inches
- 2: mm/rev or in./rev
- 3: mm/min or in./min
- 4: m/min or ft/min
- 5: Rev/min
- 6: Degrees (°)
- 7: µm or µinch

Example:

```
. . .
[//]
[la=1; s=bar diameter]
[lb=1; s=starting point in Z]
[lc=1; s=chamfer/rounding arc (-/+)]
. . .
[//]
. . .
```

Help graphics for subprogram calls

With help graphics you illustrate the calling parameters of subprograms. The CNC PILOT places the help graphics to the left next to the dialog box of the subprogram call.

As of software version 625 952-04:

If you append an underscore _ and the input field name to the name of the help graphic, the CNC PILOT will display a separate graphic for that input field. If no separate help graphic exists for an input field, the graphic for the subprogram will be displayed (if available).

Graphic format:

- BMP
- Size 410x324 pixels

You integrate help graphics for subprogram calls as follows:

- Give the help graphic the name of the subprogram (followed by the entry field name, if required) and the extension ".ico".
- Transfer the help graphic to the Data directory (on DataPilot to the machine-dependent Data directory).
- Copy the file UpHelp.res and give the copy the name of the help graphic and the extension .res. This file is likewise in the Data directory. (One res file must be provided for each graphic file.)

4.37 M Commands

M commands for program-run control

The effect of machine commands depends on the configuration of your machine. On your lathe, other M commands may apply for the listed functions. Refer to your machine manual.

Overview: M commands for program-run control	
M00	**Program stop** The program run stops. Cycle start resumes the program run.
M01	**Selectable stop** If the "Selectable stop" soft key is active in Automatic mode, the program run stops with M01. Cycle start resumes the program run. If selectable stop is not active, the program continues without stopping.
M18	**Counting pulse**
M30	**End of program** M30 indicates the end of a program or subprogram. (M30 does not need to be programmed.) If you press Cycle Start after M30, program execution is repeated from the start of the program.
M99 NS..	Program end with restart M99 means end program and start again. The CNC PILOT restarts program execution from: ■ The start of program if no NS is entered ■ The block number NS if a NS is entered
M97	Synchronous function (see "M97 Synchronous function" on page 286)



Modal functions (feed rate, spindle speed, tool number, etc.) which are effective at the end of program remain in effect when the program is restarted. You should therefore reprogram the modal functions at the start of program or at the startup block (if M99 is used).

Machine commands

The effect of machine commands depends on the configuration of your machine. The following table lists the M commands used on most machines.

M commands as machine commands	
M03	Main spindle on (cw)
M04	Main spindle on (ccw)
M05	Main spindle stop
M12	Lock main spindle brake
M13	Release main spindle brake
M14	C axis on
M15	C axis off
M19..	Spindle stop at position C
M40	Shift gear to range 0 (neutral)
M41	Shift gear to range 1
M42	Shift gear to range 2
M43	Shift gear to range 3
M44	Shift gear to range 4
Mx03	Spindle x on (cw)
Mx04	Spindle x on (ccw)
Mx05	Spindle x stop



For more information on the M commands, refer to your machine manual.

4.38 Lathes with Multiple Slides

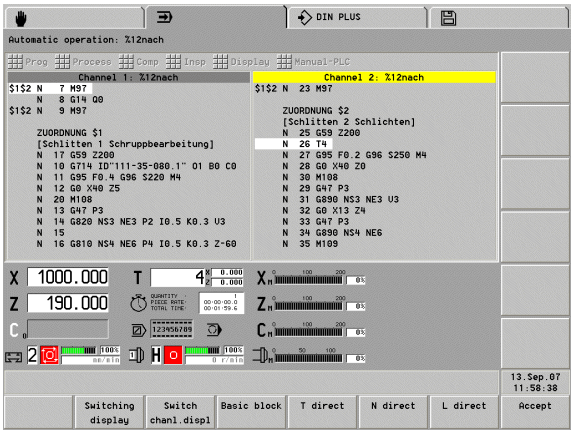
Multi-slide programming

Multiple slide programming	see:
Assignments	
Program head The "slide" input field has the following meaning: ■ No entry: The NC program is run on every slide. ■ One slide number: The NC program is run on this slide. ■ Two or more slide numbers: The NC program is run on the given slides. Enter the slide numbers in succession and without separators.	Page 136
Slide code With the slide code you assign an NC block to one or more slides: ■ NC blocks without slide designation: The NC block is executed on all slides. ■ NC blocks with slide designation: The NC block is executed on the given slides. You can program several slide designations.	Page 326
DIN PLUS word ZUORDNUNG [ASSIGNMENT] All NC commands that follow the NC block with the keyword ZUORDNUNG $x [ASSIGNMENT] (x: slide number) are assigned to the given slide. The assignment applies until a new one is programmed. If after an ASSIGNMENT you program an NC block with slide designation, the slide designation has priority.	Page 144
Reference slide for cutting speed / spindle speed For every slide that conducts an operation, a cutting speed or spindle speed must be programmed as the beginning of the program. The slide that most recently executed G96/G97 is the **reference slide.** The cutting speed / spindle speed of the reference slide applies for the machining operation. With constant surface speed (G96), the spindle speed depends on the X position. **Note:** Move to an X position that allows a sufficient cutting speed if the reference slide ends its work **before** the other slide.	Page 192
C axis on machines with multiple slides For C axes, the CNC PILOT takes into account the slide-dependent parameter "zero offset C axis 1/2" (MP 201, ..). If the slide conduct a C operation, the C axis 1 or 2 is offset. This links to the workpiece the C position that you are program. Example: On a lathe with two opposing spindles, you conduct C-axis operations with both slides. The C positions that your program are interpreted with respect to the workpiece, regardless of the slide that conducts the operation.	

Multiple slide programming	see:
End of program	
Every active slide has to execution an M30/M99 to end the NC program. We recommend programming M30/M99 without slide designation.	
Subroutines	Page 327
■ **Subprogram call:** The subprogram is called for the slide whose designation is programmed. ■ **Subprogram end:** The calling slide must conclude the subprogram with RETURN. We recommend programming the RETURN without slide designation.	
Synchronization mechanisms	
Wait for slide: Synchronous function M97	Page 286
Slides for which M97 is programmed wait until all slides in the slide designation have reached this block. Program run then continues. Either enter the slide to be synchronized in the slide designation before the M97, or program in the parameters of the M97 the slide with which it is to be synchronized.	
Simultaneous start: Synchronous start of slides G63	Page 285
G63 causes the simultaneous start of the programmed slides.	
Synchronization over marks and positions	Page 284
One-sided synchronization G62: The slide programmed with G62 waits until slide Q has reached mark H or the X/Z coordinate. If the mark **and** the X/Z coordinate are programmed, the slide waits until both conditions are fulfilled. **G162 Setting synchronizing mark:** G162 sets a synchronizing mark. The slide continues running the NC program without a break. **Note:** In a synchronization with coordinates this coordinate must be crossed over. The actual value applies. Therefore do not synchronize to the end coordinates of NC blocks, because they might not be reached, for example due to servo lag.	
Test run	Page 386
The **simulation** supports the testing of multislide axes through: ■ Display of the paths of two or more slides ■ Display of NC blocks and position values of the selected slide ■ The **synchronous point analysis** analyzes the mutual dependence of the slides. The graphic displays machining times, tool change, synchronous points and waiting times. Additional synchronous-point information show details of the selected tool change point or synchronous point.	

Program run

Block display: You can configure the block display for multiple slides. The cursor displays the active NC block for each slide.



Start-block search for multi-slide programs:

- Activate the block display for all slides involved (channels).
- Select the start block for the first slide.
- Use the slide-switch key to switch to the block display of the next slide.
- Select the start block for this slide.
- "Accept" the start blocks.
- Start the machining.

Finding a start block:

- Select a suitable start block for **each slide.**
- Every slide must have executed the same number of synchronous points by the time the start block is reached.

Positioning the steady rest

- The steady rest is positioned by subprogram.
- The workpiece is machined.
- The steady rest is moved by subprogram to a parking position.

DIN program for steady-rest positioning

%LUEN_POS.NC	
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1$2 [SLIDE]	Slide 1: tool carrier; slide 2: steady rest
. . .	
BEARBEITUNG [MACHINING]	
N 1 G59 Z1000	
. . .	
$1$2 N 2 M97	Synchronize slides 1 and 2
$2 N 3 L "LUE_POS" V1 LA300	Position the steady rest by subprogram
$1$2 N 4 M97	Slide 1 waits for steady rest
ZUORDNUNG $1 [ASSIGNMENT]	
N 5 G14 Q0	Machining with slide 1
N 6 T2	
N 7 G95 F0.6 G96 S230 M4	
N 8 G0 X350 Z10	
N 9 G810 . . .	

. . .	
$1$2 N 50 M97	Steady rest waits for end of machining
$2 N 51 L"LUE_PARK"	Steady rest by subprogram to park position
$1$2 N 52 M97	Wait until steady rest reaches park position
$1$2 N 53 M30	Program end for slides 1 and 2
ENDE [END]	

DIN subprogram for steady-rest positioning

%LUE_POS.NCS	
$2 N 1 G0 Z#__LA	Position the steady rest
$2 N 2 M300	Close the steady rest
. . .	Further steady rest commands as required
$2 RETURN	

DIN subprogram for steady-rest parking

%LUE_PARK.NCS	
$2 N 1 M301	Open the steady rest
$2 N 2 G701 Z1200	Steady rest to park position
. . .	Further steady rest commands as required
$2 RETURN	

Traveling steady rest

- The tool and the steady rest are pre-positioned (N3 to N17).
- The steady rest moves with during the cut (N19).
- After the operation, the steady rest waits until the tool has lifted off (N20 and N22).
- The steady rest is moved to a parking position (N24).

DIN program for a traveling steady-rest

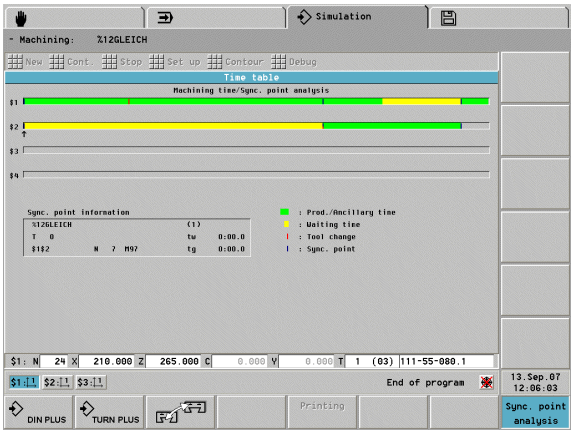
%LUENETTE.NC	
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1$2 [SLIDE]	Slide 1: tool carrier; slide 2: steady rest
. . .	
REVOLVER 1 [TURRET]	
T 2 ID"111-80-080.1"	
T 4 ID"121-55-040.1"	
. . .	
BEARBEITUNG [MACHINING]	
N 1 G59 Z1000	
. . .	
$1$2 N 2 M97	Synchronize slides 1 and 2
ZUORDNUNG $1 [ASSIGNMENT]	
N 3 G14 Q0	Slide 1: Prepare operation
N 4 T4	
N 5 G95 F0.5 G96 S200 M4	
N 6 G0 X300 Z10	
. . .	
ZUORDNUNG $2 [ASSIGNMENT]	
N 15 G0 Z10	Position the steady rest
N 16 M300	Close steady rest
N 17 G95 F0.5	Feed rate for steady rest
$1$2 N 18 G63	Slides 1 and 2 start simultaneously
$1$2 N 19 G1 Z-800	Slide 1 machines while the steady rest moves with it
ZUORDNUNG $1 [ASSIGNMENT]	
N 20 G1 X320 G162 H1	Tool lifts off and set synchronizing mark H1
N 21 G14 Q0	
ZUORDNUNG $2 [ASSIGNMENT]	
N 22 G62 H1 Q1 X318	Steady rest waits for synchronizing mark H1 and the X position 318
N 23 M301	Open the steady rest
N 24 G701 Z1200	Steady rest to park position
$1$2 N 25 M97	Wait until slides 1 and 2 have reached end position
$1$2 N 26 M30	Program end for slides 1 and 2
ENDE [END]	

Two slides work simultaneously

- In a first roughing operation the workpiece is machined to the point at which it can be recess turned.
- The recessing (N26 to N34) is conducted parallel to further roughing operations (N20 to N25)



Slide 1 defines the cutting speed. For this reason it is moved after the roughing operation to a parking position that allows an adequate cutting speed.



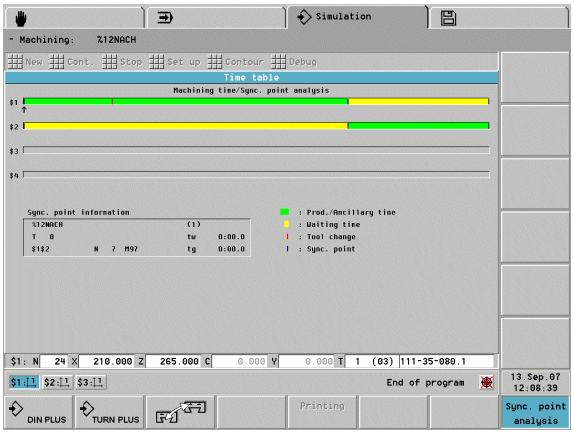
DIN program with dual-slide machining

%12GLEICH.NC	
#SCHLITTEN $1$2 [SLIDE]	
. . .	
REVOLVER 1 [TURRET]	
T 2 ID"111-80-040.1"	Roughing tool
. . .	
REVOLVER 2 [TURRET]	
T 4 ID"151-0.15-0.5"	Recessing tool
. . .	
ROHTEIL [WORKPIECE BLANK]	
N 1 G20 X30 Z80 K2	
FERTIGTEIL [FINISHED PART]	
N 2 G0 X0 Z0	
N 3 G1 X16 B-2	
N 4 G1 Z-20	
N 5 G1 X28 B1	
N 6 G1 Z-50	
N 7 G22 Z-40 II-4 K-45 B-0.5 R0.2	
. . .	
BEARBEITUNG [MACHINING]	
$1$2 N 8 M97	Synchronize slides 1 and 2
N 9 G97 S1000	
N 10 G14 Q0	Both slides move to the tool change point
$1$2 N 11 M97	Synchronize slides 1 and 2
N 12 G59 Z200	Zero point shift for both slides
. . .	
ZUORDNUNG $1 [ASSIGNMENT]	Slide 1: Roughing before recessing
N 13 T8	
N 14 G95 F0.4 G96 S220 M4	Note: G96 applies to both slides
N 15 G0 X40 Z5	
N 16 M108	
N 17 G47 P3	
N 18 G810 NS4 NE6 P2 I0.5 K0.3 X28 Z-60 W180 V3	Roughing with cutting limit
$1$2 N 19 M97	Slide 2 waits for slide 1
N 20 G47 P3	Slide 1: Further roughing

N 21 G820 NS3 NE3 P2 I0.5 K0.3 V3	
N 22 G47 P3	
N 23 G810 NS4 NE6 P4 I0.5 K0.3 Q2	
N 24 M109	
N 25 G0 X60 Z10	Slide 1: Waiting position (provides cutting speed)
ZUORDNUNG $2 [ASSIGNMENT]	Slide 2: Recessing parallel to roughing
N 26 T4	
N 27 G95 F0.2	
N 28 G0 X32 Z-44	
N 29 M108	
N 30 G47 P3	
N 31 G866 NS7 I0.2	
N 32 G0 X32 Z-44	
N 33 M109	
N 34 G14 Q0	Slide 2: Move to the tool change position
$1$2 N 35 M97	Slide 1 waits for slide 2
$1 N 36 G14 Q0	Slide 1: Move to the tool change position
$1$2 N 37 M30	Program end for slides 1 and 2
ENDE [END]	

Two slides work in succession

- Slide 1 executes the roughing operation (N10 to N20).
- Then slide 2 finishes the contour (N22 to N34).



DIN program with two consecutive slides

%12NACH.NC	
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1$2 [SLIDE]	
. . .	
REVOLVER 1 [TURRET]	
T 2 ID"111-80-040.1"	Roughing tool
. . .	
T 4 ID"121-55-040.1"	Finishing tool
. . .	
N 1 G20 X30 Z80 K2	
FERTIGTEIL [FINISHED PART]	
N 2 G0 X0 Z0	
N 3 G1 X16 B-2	
N 4 G1 Z-20	
N 5 G1 X28 B1	
N 6 G1 Z-50	
. . .	
BEARBEITUNG [MACHINING]	
$1$2 N 7 M97	Synchronize slides 1 and 2
N 8 G14 Q0	Both slides move to the tool change point
$1$2 N 9 M97	Synchronize slides 1 and 2
. . .	
ZUORDNUNG $1 [ASSIGNMENT]	Slide 1: Roughing
N 10 G59 Z200	
N 11 T8	
N 12 G95 F0.4 G96 S220 M4	
N 13 G0 X40 Z5	
N 14 M108	
N 15 G47 P3	
N 16 G820 NS3 NE3 P2 I0.5 K0.3 V3	
N 17 G810 NS4 NE6 P4 I0.5 K0.3 Z-60 W180 Q2	
N 18 M109	
N 19 G0 X60 Z10	
N 20 G14 Q0	

$1$2 N 21 M97	Slide 2 waits for slide 1
ZUORDNUNG $2 [ASSIGNMENT]	Slide 2: Finishing
N 22 G59 Z200	
N 23 T4	
N 24 G95 F0.2 G96 S250 M4	
N 25 G0 X40 Z0	
N 26 M108	
N 27 G47 P3	
N 28 G890 NS3 NE3 V3	
N 29 G0 X13 Z4	
N 30 G47 P3	
N 31 G890 NS4 NE6	
N 32 M109	
N 33 G0 X60 Z10	
N 34 G14 Q0	
$1$2 N 35 M97	Synchronize slides 1 and 2
$1$2 N 36 M30	Program end for slides 1 and 2
ENDE [END]	

Machining with four-axis cycle

- Slides 1 and 2 execute the roughing operation together (N8 to N15). During this, the roughing cycle G810 is used as four-axis cycle.
- Then slide 1 finishes the contour (N16 to N18).

DIN program with four-axis machining

%4ACHS.NC	
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1$2 [SLIDE]	
. . .	
REVOLVER 1 [TURRET]	
T 1 ID"111-80-080.1"	Roughing tool
T 2 ID"121-55-040.1"	Finishing tool
. . .	
REVOLVER 2 [TURRET]	
T 1 ID"111-80-040.1"	Roughing tool
. . .	
ROHTEIL [WORKPIECE BLANK]	
N 1 G20 X100 Z200 K0	

FERTIGTEIL [FINISHED PART]	
N 2 G0 X0 Z0	
N 3 G1 X50 B8	
N 4 G1 Z-150 B6	
N 5 G1 X100 B5	
N 6 G1 Z-200	
. . .	
BEARBEITUNG [MACHINING]	
$1$2 N 7 M97	Synchronize slides 1 and 2
ZUORDNUNG $1$2 [ASSIGNMENT]	Both slides: Tool change and pre-positioning
N 8 G14 Q0	
N 9 T1	
N 10 G59 Z300	
N 11 G0 X120 Z5 G95 F1	
$1$2 N 12 M97	Synchronize slides 1 and 2
$1 N 13 G96 S300 M4	
N 14 G810 NS4 NE5 P5 I0.5 K0.4 B0	Slides 1 and 2 rough simultaneously
N 15 G14	
ZUORDNUNG $1 [ASSIGNMENT]	Slide 1: Finishing
N 16 T2	
N 17 G890 NS4 NE5	
N 18 G14	
$1$2 N 19 M97	Synchronize slides 1 and 2
$1$2 N 20 M30	Program end for slides 1 and 2
ENDE [ENDE]	

4.39 Full-surface machining

Fundamentals of full-surface machining

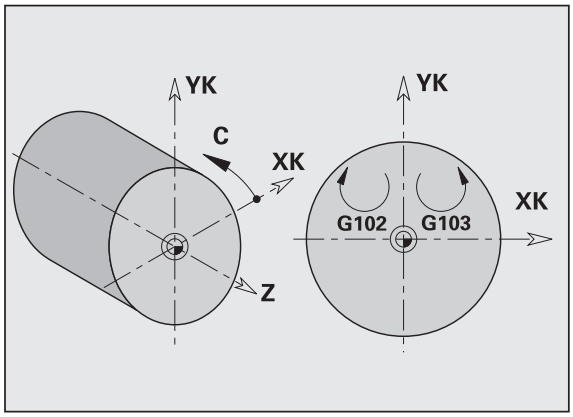
In "full-surface machining," the front and rear ends can be machined in **one** NC program. The CNC PILOT supports full-surface machining for all common machine designs. The features include angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation. This ensures efficient full-surface machining and simple programming.

You describe the turning contour, the contours for the C axis as well as full-surface machining functions in one NC program. Expert programs are available for configuring the lathe.

You can also enjoy the benefits of full-surface machining on lathes with one main spindle.

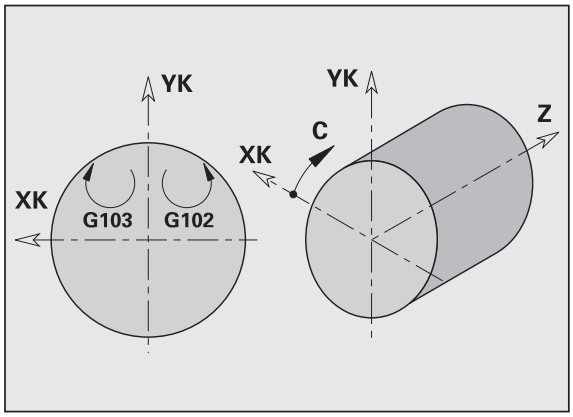
Rear-face contours with C axis: The XK axis and therefore also the C axis are oriented with respect to the workpiece, not to the spindle. Therefore, for the rear face:

- Orientation of the XK axis: To the left (front face: to the right)
- Orientation of the C axis: Clockwise
- Direction of rotation for circular arcs G102: Counterclockwise.
- Direction of rotation for circular arcs G103: Clockwise.



Turning: The CNC PILOT supports full-surface machining with conversion and mirroring functions so that rear-face machining also follows these principles:

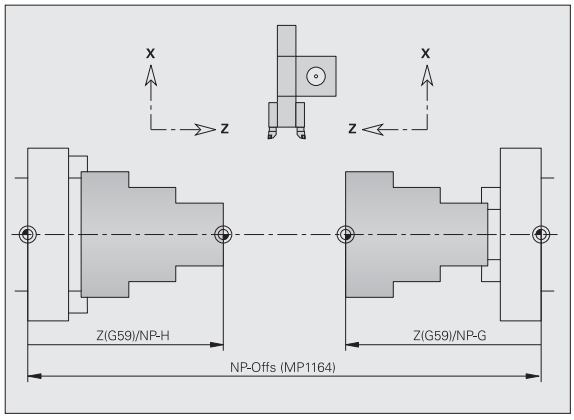
- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.



The machine manufacturer usually supplies your lathe with suitable **expert programs** for workpiece transfer.

Reference points and coordinate system: The position of the machine and workpiece zero points as well as the coordinate systems for the spindle and opposing spindle are illustrated in the figure at bottom. With this design of lathe it is recommended to mirror only the Z axis. Then, for either spindle, moving in positive direction will stand for motion away from the workpiece.

Usually the expert program contains the mirroring of the Z axis and the zero-point shift by the dimension “NP-Offs”.



Programming of full-surface machining

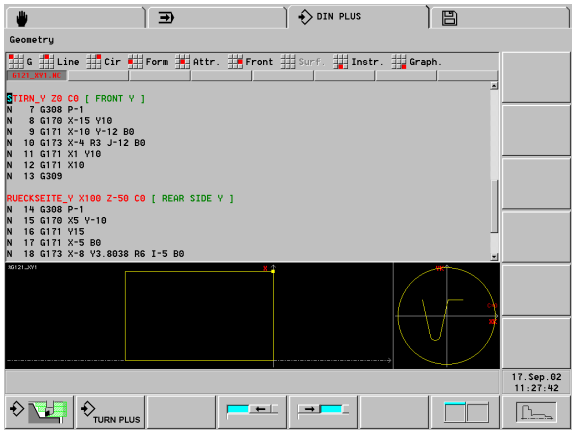
When programming a contour on the rear face, be sure to consider the orientation of the XK axis (or X axis) and rotational direction of arcs.

Insofar as you use drilling and milling cycles, there are no special aspects to rear-face machining, since these cycles refer to predefined contours.

For rear-face machining with the basic commands G100 to G103 the same conditions apply as for rear-face contours.

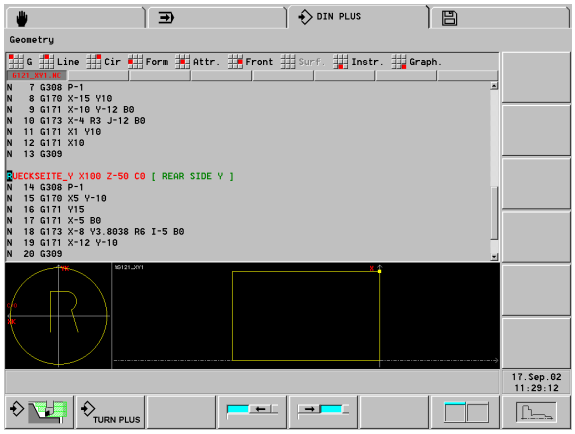
Turning operations: The expert programs for reclamping contain converting and mirroring functions. The following principle applies for rear-face machining (2nd setup):

- \+ direction: Goes away from the workpiece
- – direction: Goes toward the workpiece
- G2/G12: Circular arc clockwise
- G3/G13: Circular arc counterclockwise



Working without expert programs
If you do not use the expert programs or the converting and mirroring functions, the following principle applies:

- **+ direction:** Goes away from the spindle
- **– direction:** Goes toward the main spindle
- **G2/G12:** Circular arc clockwise
- **G3/G13:** Circular arc counterclockwise



Full-surface machining with opposing spindle

G30: The expert program activates the mirroring of the Z axis and the conversion of the arcs (G2, G3, ..). The arcs must be converted for turning operations and operations with the C axis.

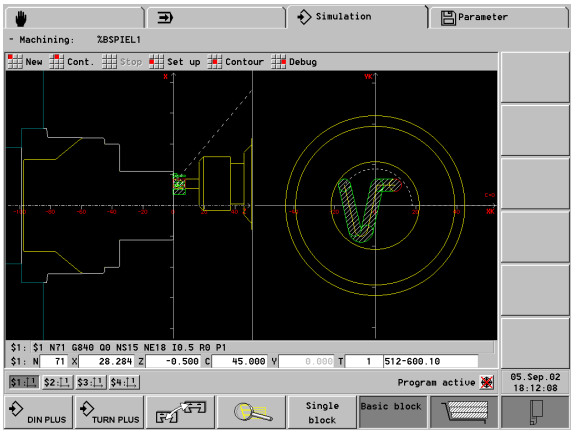
G121: The expert program shifts the contour and mirrors the coordinate system (Z axis). Further programming of G121 is normally not required for machining the rear face after rechucking.

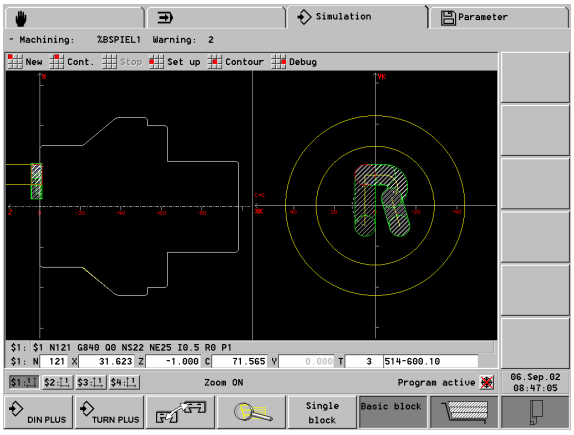
Example: The workpiece is machined on the front face, transferred to the opposing spindle through an expert program and machined on the rear face (see illustrations).

The expert program is used for:

- Angle-synchronous workpiece transfer to the opposing spindle
- Mirroring traverse paths in the Z axis
- Activating a conversion list
- Mirroring the contour description and shifting for the 2nd chucking

The mirroring/conversion function for rear-face machining (expert program), is switched off at program end with the G30 command.





Full-surface machining on machines with opposing spindles

PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1$2 [SLIDE]	
. . .	
REVOLVER 1 [TURRET]	
T1 ID "512-600.10"	
T2 ID "111-80-080.1"	
T3 ID "514-600.10"	
T4 ID "121-55-040.1"	
T6 ID "115-80.080"	
T8 ID "125-55.040"	
SPANNMITTEL 1 [CHUCKING EQUIPMENT] (ZERO POINT SHIFT Z233)	Chucking equipment for 1st setup
H1 ID"3BACK"	
H2 ID"KBA250-86" X100 Q4	
SPANNMITTEL 4 [CHUCKING EQUIPMENT] (ZERO POINT SHIFT Z196)	Chucking equipment for 2nd setup
H1 ID"3BACK"	
H2 ID"WBA240-50" X80 Q4	
ROHTEIL [WORKPIECE BLANK]	
N1 G20 X100 Z100 K1	
FINISHED PART	
. . .	
STIRN Z0 [FRONT]	
N13 G308 P-1	
N14 G100 XK-15 YK10	
N15 G101 XK-10 YK12 B0	
N16 G103 XK-4.0725 YK-12.6555 R3 J-12	
N17 G101 XK1 YK10	
N18 G101 XK10	
N19 G309	
RUECKSEITE Z-98 [REAR SIDE]	
. . .	
BEARBEITUNG [MACHINING]	
N27 G59 Z233	Zero point shift for 1st setup
$1 N28 G65 H1 X0 Z-135 D1	Display chucking equipment of 1st setup

$1 N29 G65 H2 X100 Z-99 D1 Q4	
$1 N30 G14 Q0	
$1 N31 G26 S2500	
$1 N32 T2	
. . .	
$1 N62 G126 S4000	Milling – contour – outside – front face
$1 N63 M5	
$1 N64 T1	
$1 N65 G197 S1485 G193 F0.05 M103	
$1 N66 M14	
$1 N67 M107	
$1 N68 G0 X36.0555 Z3	
$1 N69 G110 C146.31	
$1 N70 G147 I2 K2	
$1 N71 G840 Q0 NS15 NE18 I0.5 R0 P1	
$1 N72 G0 X31.241 Z3	
$1 N73 G14 Q0	
$1 N74 M105	
$1 N75 M109	
$1 N76 M15	Prepare the rechucking
$1 N77 G65 H1 D1	Delete chucking equipment of 1st setup
$1 N78 G65 H2 D1	
$1 $2 N79 M97	Synchronize slides for rechucking procedure
$1 $2 N80 L"UMKOMPL" V1 LA1000 LD369 LE547 LF98 LH98 I3	Expert program for parting and rechucking procedures: LA=speed limitation LD=transfer position Z LE=machining position Z – slide 2 LF=length of finished part LH=distance from chuck reference point to edge of workpiece I=minimum feed motion to fixed stop
$1 $2 N81 M97	
$1 N82 G65 H1 X0 Z-100 D4	Switch on chucking equipment for spindle 4
$1 N83 G65 H2 X80 Z-63 D4 Q4	
. . .	Rear-side machining
$1 $2 N125 G30 H0 Q0	Switch off rear-face machining
$1 $2 N126 M97	
N129 M30	
ENDE [END]	

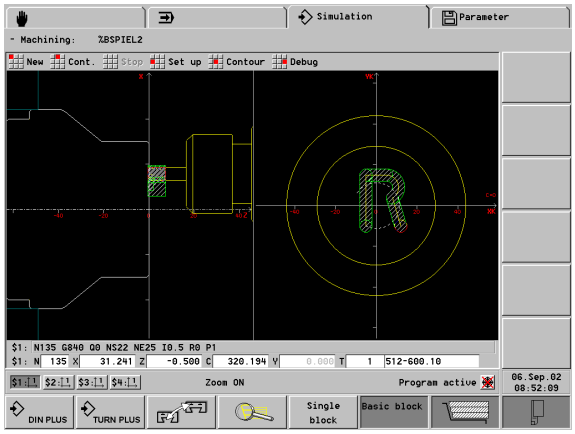
Full-surface machining with single spindle

G30: Normally not required

G121: The expert program mirrors the contour. Further programming of G121 is normally not required for machining the rear face after rechucking.

Example: Describes the machining of the front and rear face, using **one** NC program. The workpiece is first machined on the front face; then it is rechucked manually. The rear face is machined subsequently.

The expert program mirrors and shifts the contour for the 2nd setup.



Full-surface machining on machine with one spindle

```
PROGRAMMKOPF [PROGRAM HEAD]
#SCHLITTEN $1 [SLIDE]
REVOLVER 1 [TURRET]
T1 ID "512-600.10"
T2 ID "111-80-080.1"
T4 ID "121-55-040.1"

SPANNMITTEL 1 [CHUCKING EQUIPMENT] (ZERO POINT SHIFT
Z233)
H1 ID"3BACK"
H2 ID"KBA250-86" X100 Q4

ROHTEIL [WORKPIECE BLANK]
N1 G20 X100 Z100 K1

FERTIGTEIL [FINISHED PART]
. . .
STIRN Z0 [FRONT]
. . .
RUECKSEITE Z-98 [REAR SIDE]
N20 G308 P-1
N21 G100 XK5 YK-10
N22 G101 YK15
N23 G101 XK-5
```

N24 G103 XK-8 YK3.8038 R6 I-5 B0	
N25 G101 XK-12 YK-10	
N26 G309	
BEARBEITUNG [MACHINING]	
N27 G59 Z233	Zero point shift for 1st setup
N28 G65 H1 X0 Z-135 D1	Display chucking equipment of 1st setup
N29 G65 H2 X100 Z-99 D1 Q4	
. . .	
N82 M15	Prepare the rechucking
N83 G65 H1 D1	Delete chucking equipment of 1st set-up
N84 G65 H2 D1	
N86 L"UMHAND" V1 LF98 LH99	Expert program for manual rechucking: LF=length of finished part LH=distance from chuck reference point to edge of workpiece
N88 G65 H1 X0 Z-99 D1	Switch on chucking equipment for rear-face machining
N89 G65 H2 X88 Z-63 D1 Q4	
. . .	
N125 M5	Milling – rear face
N126 T1	
N127 G197 S1485 G193 F0.05 M103	
N128 M14	
N130 M107	
N131 G0 X22.3607 Z3	
N132 G110 C-116.565	
N133 G153	
N134 G147 I2 K2	
N135 G840 Q0 NS22 NE25 I0.5 R0 P1	
N136 G0 X154 Z-95	
N137 G0 X154 Z3	
N138 G14 Q0	
N139 M105	
N141 M109	
N142 M15	
N143 M30	
ENDE [END]	

4.40 DIN PLUS Program Example

Example of a subprogram with contour repetitions

Contour repetitions, including saving of the contour

PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN $1 [SLIDE]	
REVOLVER 1 [TURRET]	
T2 ID "121-55-040.1"	
T3 ID "111-55.080.1"	
T4 ID "161-400.2"	
T8 ID "342-18.0-70"	
T12 ID "112-12-050.1"	
BLANK	
N1 G20 X100 Z120 K1	
FERTIGTEIL [FINISHED PART]	
N2 G0 X19.2 Z-10	
N3 G1 Z-8.5 B0.35	
N4 G1 X38 B3	
N5 G1 Z-3.05 B0.2	
N6 G1 X42 B0.5	
N7 G1 Z0 B0.2	
N8 G1 X66 B0.5	
N9 G1 Z-10 B0.5	
N10 G1 X19.2 B0.5	
BEARBEITUNG [MACHINING]	
N11 G26 S2500	
N12 G14 Q0	
N13 G702 Q0	Save contour
N14 L"1" V0 Q2	Qx = number of repetitions
N15 M30	
UNTERPROGRAMM "1" [SUBPROGRAM]	
N16 M108	
N17 G702 Q1	Load saved contour

N18 G14 Q0	
N19 T8	
N20 G97 S2000 M3	
N21 G95 F0.2	
N22 G0 X0 Z4	
N23 G147 K1	
N24 G74 Z-15 P72 I8 B20 J36 E0.1 K0	
N25 G14 Q0	
N26 T3	
N27 G96 S300 G95 F0.35 M4	
N28 G0 X72 Z2	
N29 G820 NS8 NE8 P2 K0.2 W270 V3	
N30 G14 Q0	
N31 T12	
N32 G96 S250 G95 F0.22	
N33 G810 NS7 NE3 P2 I0.2 K0.1 Z-12 H0 W180 Q0	
N34 G14 Q2	
N35 T2	
N36 G96 S300 G95 F0.08	
N37 G0 X69 Z2	
N38 G47 P1	
N39 G890 NS8 V3 H3 Z-40 D3	
N40 G47 P1	
N41 G890 NS9 V1 H0 Z-40 D1 I74 K0	
N42 G14 Q0	
N43 T12	
N44 G0 X44 Z2	
N45 G890 NS7 NE3	
N46 G14 Q2	
N47 T4	Insert parting tool
N48 G96 S160 G95 F0.18 M4	
N49 G0 X72 Z-14	
N50 G150	Shift reference point to the right of the cutting edge
N51 G1 X60	
N52 G1 X72	
N53 G0 Z-9	
N54 G1 X66 G95 F0.18	
N55 G42	Activate TRC

N56 G1 Z-10 B0.5	
N57 G1 X17	
N58 G0 X72	
N59 G0 X80 Z-10 G40	Turn TRC off
N60 G14 Q0	
N61 G56 Z-14.4	Incremental zero point shift
RETURN	
ENDE [END]	

4.41 DIN PLUS Templates

A template is a predefined NC code block that is adapted to your lathe and is integrated in the NC program. That reduces the programming effort while helping you to standardize.

The CNC PILOT differentiates between:

- The **starting template** for making a new NC program.
- **Structure templates**, which support you when programming complex processes.

The templates are saved in the NCPS directory under the name DINSTART.BEV or VORLAGEx.BEV (x: 1 to 9).

Starting templates

If a **starting template** is available, the CNC PILOT loads it when you make a new NC program.

The starting template should contain program section codes, constant definitions, speed limitations, zero point shifts, and other instructions and data that are adapted to your lathe. If no starting template exists, the CNC PILOT creates a new NC program containing only the standard program section codes.

To edit the starting template:

- Log on as System Manager.
- Select "Prog > Load > Template" in the main menu.
- Choose DINSTART from the template list.
- Edit the template using the free editing function and then save.

If there is no starting template on your control, you can make a template using an external application. Name the template DINSTART.BEV and copy it to the NCPS directory.

Structure templates

A structure template defines program sequences that are loaded into the NC program when the template is called. You can influence a structure template by using transfer parameters. This makes it easier to program complex lathes.

Structure templates are usually provided by the machine tool builder, together with a description of their functionality. The CNC PILOT supports up to 9 structure templates.

To call a **structure template**:

- Select "Instr. < Template selection < .." in the machining menu (".." the last level of the template menu depends on the specific lathe).

Design of structure templates

When you call a structure template, the NC blocks of the template are loaded into the NC program. The blocks of the template have been designed in such a way that you can influence the NC program by adding or omitting entries. This is done by using transfer parameters. In addition, the CNC PILOT adds the block numbers.

To influence a structure template:

- **Placeholder:** Placeholders in the template have the following syntax: #__la (or any other parameter designation). The placeholder will then be replaced with the transfer data "la" (or any other parameter designation). The transfer data can be any simple text, an M or T function or a G function call (including the parameters). The transfer data type is defined in the declaration of the transfer parameter.
- **Omit line:** If you want the CNC PILOT to omit NC blocks, the relevant NC blocks must be preceded by [[#__la]] (or any other parameter designation) in the template. The corresponding transfer parameter la (or any other parameter designation) is declared using the "yes/no decision" transfer type. The NC block concerned is only transferred to the program if the condition is true, i.e. yes was entered.

Transfer parameters for structure templates

The CNC PILOT supports up to 19 transfer parameters:

- [//] – Beginning of the parameter declaration
- [pn; s=dialog text (up to 16 characters); xx]
- [//] – End of the parameter declaration

pn: Parameter designations (la, lb, ...)

xx: Type of data transfer:

- No type defined: The entered text is used
- e=S0: yes/no decision, default: no
- e=S1: yes/no decision, default: yes
- e=G: G function
  - After you enter the G number, the CNC PILOT opens the dialog box for this G function. The G function call is transferred including its parameters.
  - When you press Continue, the list of G functions is presented for selection.
- e=M: M function
  - Enter the M number. The M function call is transferred.
  - When you press Continue, the list of M functions is presented for selection.
- e=T: The CNC PILOT opens the turret list for selecting a tool. The tool call selected in the turret list is transferred.

Example: Transfer parameters

```
Vorlagex.BEV
[//]
[/la;   s=spindle 0   ;e=S0/]
[/lb;   s=G function  ;e=G/]
[/lc;   s=M function  ;e=M/]
[/ld;   s=T function  ;e=T/]
[/le;   s=SP name  /]
[//]
. . .
```

Editing structure templates

- Log in as "System Manager"
- Select "Prog > Load > Template" in the main menu.
- Choose Vorlagex from the template list.
- Edit the template using the free editing function and then save.

Help graphics for structure templates

Help graphics illustrate the transfer parameters of structure templates. The CNC PILOT places the help graphics to the left next to the dialog box.

The help graphic is named after the template it illustrates. If you append an underscore _ and the input field name to the name of the help graphic, the CNC PILOT will display a separate graphic for that input field. If no separate help graphic exists for an input field, the graphic for the template will be displayed (if available).

Graphic format:

- BMP graphics
- Size 410x324 pixels

You integrate help graphics for templates as follows:

- Give the help graphic the name of the template (followed by the entry field name, if applicable) and the extension ".ico".
- Transfer the help graphics to the Data directory (on DataPilot to the machine-dependent Data directory).
- Copy the file UpHelp.res and give the copy the name of the help graphic and the extension .res. This file is likewise in the Data directory. (One res file must be provided for each graphic file.)

Template menu

Template menu: The last level of the template menu is defined using the language dependent fixed-word list "...". Enter the menu text for templates 1 to 9 in this fixed-word list.

Template example

Example VORLAGEx.BEV

%VORLAGEX.BEV	Work block for slide 1
[//]	Declare transfer parameters
[/LB; S=TOOL TO SPINDLE0 ;E=S0/]	yes/no decision
[/LC; S=TOOL TO SPINDLE3 ;E=S0/]	
[/LF; S=G FUNCTION ;E=G/]	G function
[/LH; S=MAKE SP ;E=S0/]	
[/J; S=SP NAME /]	Transfer the entered text
[//]	
[[#__LH]] [===== SUBPROGRAM ====]	
[[#__LH]] UNTERPROGRAMM "#__J" [SUBPROGRAM]	
[[#__LB]] G714 ID "" [TOOL]	Slide 1 to spindle 0
[[#__LB]] G96 S100 G95 F0.05 M4 [TECHNOLOGY]	Technology data for main spindle
[[#__LB]] G0 [APPROACH POSITION]	
[[#__LB]] M107 [COOLANT ON]	
[[#__LB]] G47 P3 [SAFETY CLEARANCE]	
[[#__LB]] #__LF	Placeholder for G function
[[#__LB]] M109 [COOLANT OFF]	
[[#__LB]] G14 Q1 [MOVE TO THE TOOL CHANGE POSITION]	
[[#__LC]] G714 ID "" [TOOL]	Slide 1 to spindle 3
[[#__LC]] G396 S100 G395 F0.05 M303 [TECHNOLOGY]	Technology data for spindle 3
[[#__LC]] G0 [APPROACH POSITION]	
[[#__LC]] M107 [COOLANT ON]	
[[#__LC]] G47 P3 [SAFETY CLEARANCE]	
[[#__LC]] #__LF	Placeholder for G function
[[#__LC]] M109 [COOLANT OFF]	
[[#__LC]] G14 Q1 [MOVE TO THE TOOL CHANGE POSITION]	
[[#__LH]] RETURN	

A **template call** is programmed by entering:

- Tool to spindle0: no
- Tool to spindle3: yes
- G function: 810 and the parameters of G810
- Make SP: yes
- SP name: Schru1

From this data, the CNC PILOT generates the following program sequence:

[===== SUBPROGRAM ====]	
UNTERPROGRAMM "SCHRU1" [SUBPROGRAM]	Call the subprogram with the entered SP name
N 2 G714 ID "" [TOOL]	Slide 1 to spindle 3
N 3 G396 S100 G395 F0.05 M303 [TECHNOLOGY]	
N 4 G0 [APPROACH POSITION]	
N 5 M107 [COOLANT ON]	
N 6 G47 P3 [SAFETY CLEARANCE]	
N 7 G810 NS.. NE.. ...	G function with entered parameters
N 8 M109 [COOLANT OFF]	
N 9 G14 Q1 [MOVE TO THE TOOL CHANGE POSITION]	
RETURN	

4.42 Connection between Geometry and Machining Commands

Turning

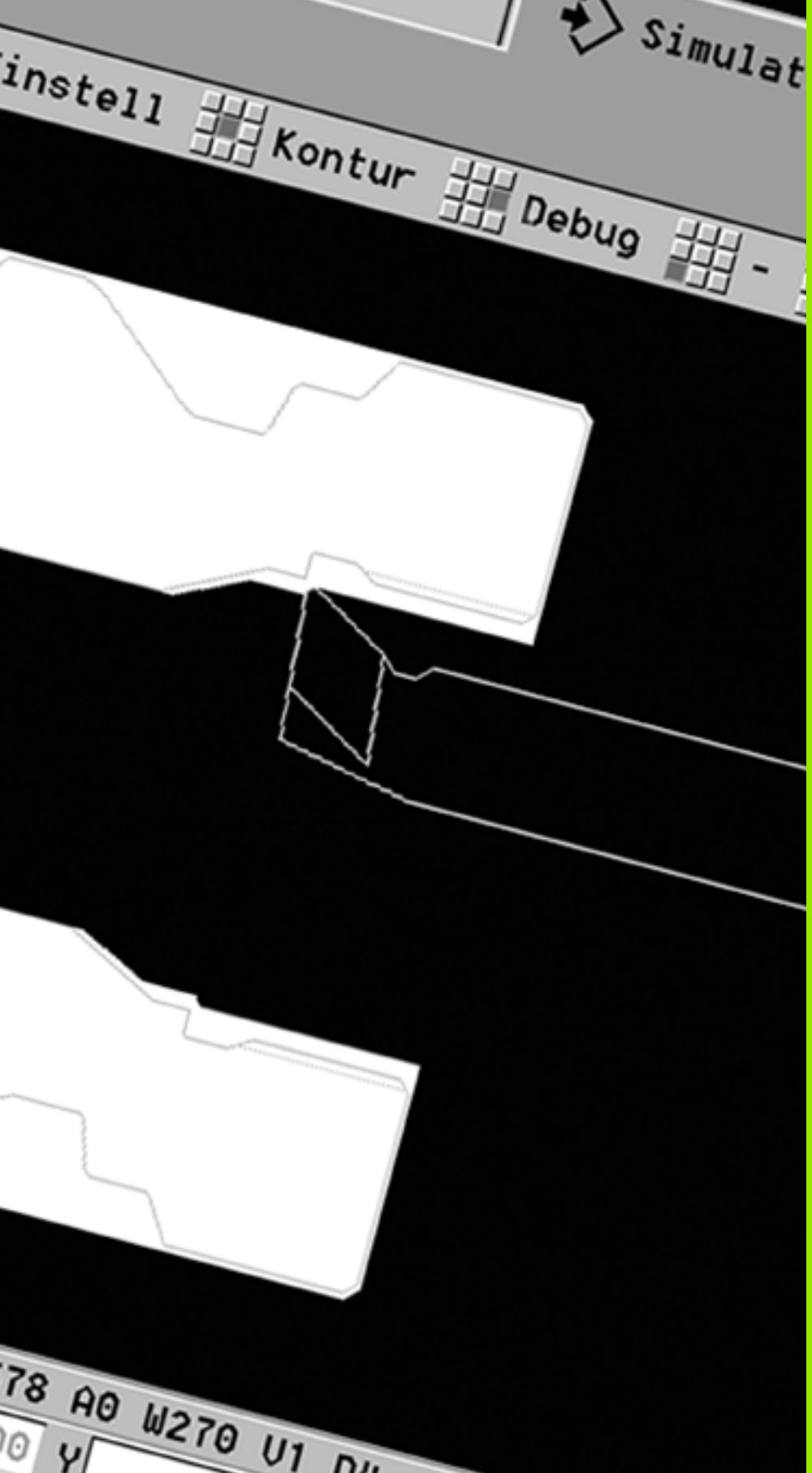
Function	Geometry	Machining operation
Individual elements	■ G0 to G3 ■ G12/G13	■ G810 Longitudinal roughing cycle ■ G820 Transverse roughing cycle ■ G830 Contour-parallel roughing cycle ■ G835 Contour-parallel with neutral tool ■ G860 Universal recessing cycle ■ G869 Recess turning cycle ■ G890 Finishing cycle
Recess	■ G22 (standard)	■ G860 Universal recessing cycle ■ G866 Simple recessing cycle ■ G869 Recess turning cycle
Recess	■ G23	■ G860 Universal recessing cycle ■ G869 Recess turning cycle
Thread with undercut	■ G24	■ G810 Longitudinal roughing cycle ■ G820 Face roughing cycle ■ G830 Contour-parallel roughing cycle ■ G890 Finishing cycle ■ G31 Thread cycle
Undercut	■ G25	■ G810 Longitudinal roughing cycle ■ G890 Finishing cycle
Threads	■ G34 (standard) ■ G37 (general)	■ G31 Thread cycle
Hole	■ G49 (turning center)	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle

C-axis machining – front/rear face

Function	Geometry	Machining
Individual elements	■ G100 to G103	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Figures	■ G301 Linear groove ■ G302/G303 Circular slot ■ G304 Full circle ■ G305 Rectangle ■ G307 Eccentric polygon	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Hole	■ G300	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle

C-axis machining – lateral surface

Function	Geometry	Machining
Individual elements	■ G110 to G113	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Figures	■ G311 Linear groove ■ G312/G313 Circular slot ■ G314 Full circle ■ G315 Rectangle ■ G317 Eccentric polygon	■ G840 Contour milling ■ G845/G846 Pocket milling, roughing/finishing
Hole	■ G310	■ G71 Simple drilling cycle ■ G72 Counterboring, countersinking, etc. ■ G73 Tapping cycle ■ G74 Deep hole drilling cycle



5

Graphic Simulation

5.1 Simulation Mode of Operation

The Simulation mode shows a graphic representation of programmed contours, the paths of traverse and cutting operations. The CNC PILOT shows the working space, tools and chucking equipment true to scale.

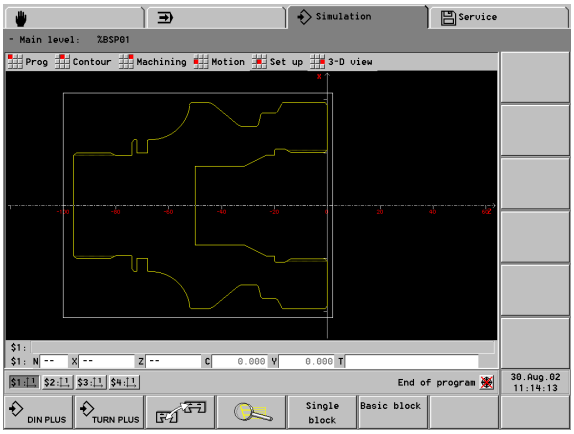
Check machining operations with the C axis in the supplementary windows (front/surface and side view windows).

For complex NC programs with branches, variable calculations, external events, etc., you simulate all inputs and events to test all program branches.

The CNC PILOT supports the program text for lathes with **more than one slide** and **up to four workpieces** in one working space.

During simulation, the CNC PILOT calculates the **machining and idle-machine times** for every tool.

The **synchronous point analysis** support the the analysis and optimization of NC programs for multiple slides.

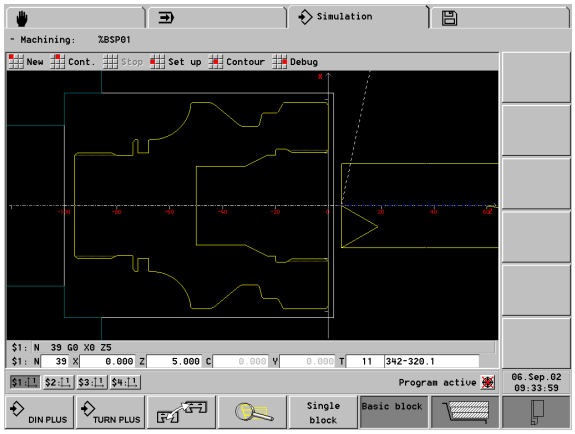


Functions of the machining simulation:

- **Contour simulation:** Simulation of programmed contours (see “Contour Simulation” on page 374)
- **Machining simulation:** Checking the machining process (see “Machining Simulation” on page 376)
- **Motion simulation:** Simulation of real-time machining with continuous contour regeneration (see “Motion Simulation” on page 380)
- **3-D view:** 3-D depiction of turned contours (see “3-D View” on page 382)
- **Time calculation:** Display of the machining times and idle times for each tool used (see “Time Calculation” on page 387)
- **Synchronous point analysis:** Depiction of workpiece machining with multiple slides. The display shows both the time sequence and the dependencies of the slides among each other (see “Synchronous point analysis” on page 388).
- **Debug functions:** Display and simulation of variables and events (see “Debugging Functions” on page 383)

Screen layout, soft keys

Screen layout

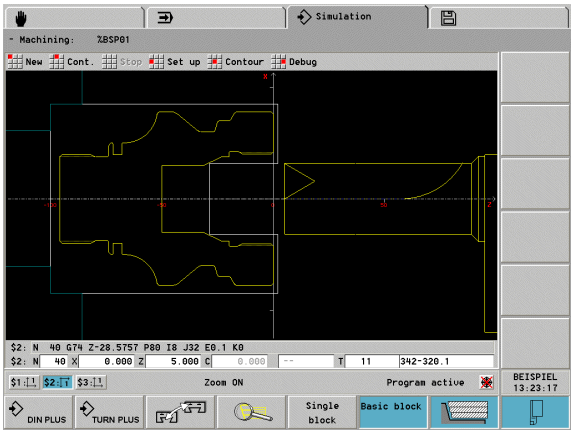


1 Info line: Submode of simulation, simulated NC program
2 Simulation window: The machining is depicted in up to three windows
3 Block display: Programmed NC block or, alternatively, display of variables
4 Position display: NC block number, position values, tool information or, alternatively, cutting values
5 Slide symbols
6 Status of the simulation, status of the zero point shift

Soft keys

DIN PLUS	Change to the DIN PLUS mode
TURN PLUS	Change to the TURN PLUS mode
	Switch to the next slide
	Activate the magnify function
Single block	Single block mode: Stop after every NC source block
Basic block	Basic block mode: Stops after every contour element or after every traverse path
	View of the paths of traverse: **Line** or **(cutting) track**
	Tool depiction: **Point of light** or **tool**
	Changes to next simulation window
	Call the next selection in dialogs

Graphic elements



- **Coordinate systems:** The zero point of the coordinate system corresponds to the workpiece zero point. The arrows of the X and Z axes point in the positive direction. If the NC program is machining more than one workpieces, the coordinate systems of all required slides are displayed.
- **Workpiece blank depiction**
  - Programmed: Programmed blank
  - Not programmed: standard blank from control parameter 23.
- **Finished part depiction (and auxiliary contours)**
  - Programmed: Programmed finished part
  - Not programmed: No display
- **Tilted plane:** The simulation depicts the tilted plane as auxiliary contour, if it is programmed with SURFACE_Y ...
- **Tool display.** The CNC PILOT uses the parameters of the tool database to display a tool. Whether the complete tool or only the cutting area is shown is specified in “Picture number” (picture no.=–1: No tool displayed).
  - In the NC program, the tool programmed in the TURRET section is used
  - Not programmed in the NC program: The entry from the tool list is used
- **Chucking equipment depiction:** The simulation graphics depict the chucking equipment provided that it has been programmed with G65.
  The CNC PILOT uses the parameters of the chucking equipment database to depict a chuck.
- **Light dot:** The light dot (small white square) represents the theoretical tool tip.

Displays

The **block display** displays the programmed NC blocks (NC source blocks). In the “Window selection” dialog box you specify (see “Simulation window” on page 368):

- Block display for the specified slides
- Block display for the slides marked in the “Window selection” dialog box

As an alternative to the block display the simulation displays four variables: see “Debugging Functions” on page 383

Position display:

The following boxes of the display cannot be edited:

- **N:** Block number of the NC source block
- **X, Z, C:** Position values (actual values)

The following display fields depend on the status line:

- Default settings (values of the selected slide):
  - Position values (actual values)
  - Turret pocket of the active tool
- Technology data settings:
  - Shaft speed
  - Feed rate
  - Direction of spindle rotation

Switching between standard setting and display of technology data

▶ Select “Set up > Status line” or press PgUp/PgDn.



The control parameter 1 (settings) defines whether the display values will be in millimeters or inches. The setting in PROGRAM HEAD has no influence on operation and display in the simulation mode of operation.

Displays for slides: The slide symbols contain information on the coordinate system and the contour currently being machined.

Slide symbols



Information of the slide symbols:

- $n (n: 1..6): Slide designation
- Configured coordinate system
- Number in the coordinate system: Contour that this slide is currently machining
- The symbol of the selected slide is marked



You can switch slides by soft key.

Displays for contours: If several contours are defined in the NC program, the simulation displays the corresponding contour symbols.

Contour symbols



Information of the contour symbols:

- Qn (n: 1..4): Contour n
- Position of the coordinate system
- The symbol of the selected contour is marked

The simulation window displays the coordinate system of the selected contour.

Selection of a contour

- Select "Setup > Contour selection." The simulation opens the "Contour selection" dialog box.
- In the "Contour selection" field, set the desired contour

Zero point shifts

In the "Contour selection" dialog box (Set up > Contour selection) you define whether zero point shifts will be accounted for in the simulation. As an alternative you can use the touchpad to click the zero-point-shifts symbol in order to change the setting.

If you use the program section code **CONTOUR** and **G99,** no matter what the status of the zero point shift:

- The workpiece (the contour) is depicted at the position defined in CONTOUR
- G99 X.. Z.. shifts the workpiece to a new position



Including zero point shifts

- The machine zero point is the reference point for the positioning of contours and for the traverse paths.
- Zero point shifts are included in calculation



Zero point shifts are **not included** in calculation:

- The workpiece zero point is the reference point for the paths of traverse
- Zero point shifts are ignored



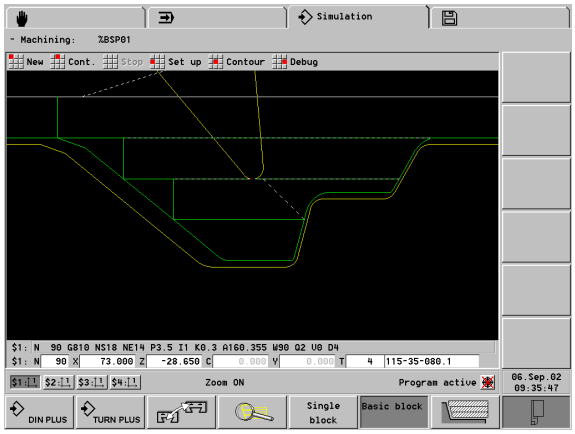
A change of status is not accounted for until the simulation is restarted. The symbols are shown faded as long as the changed setting is not yet in effect.

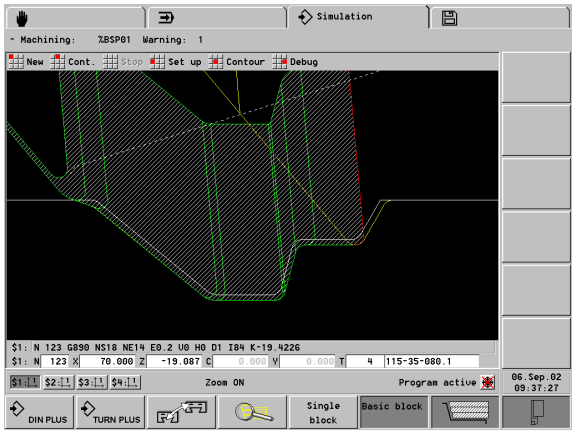
Path display

Rapid paths are shown as a broken white line.

Feed paths are displayed either as a line or as a cutting trace, depending on the soft-key setting:

- **Line display:** A solid line describes the path of the theoretical tool tip (wire-frame graphics) The wire frame display is particularly convenient if you only need a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This graphics are therefore not as suitable if you wish to run a thorough check on the machined contour. In the CNC, this “falsification” is compensated by the cutting radius compensation. In control parameter 24, you can define the color of the feed path for the respective T number.
- **Cutting path display:** The CNC PILOT uses hatch marking to depict the surface covered by the cutting area of the tool. The cutting path graphic accounts for the exact geometry of the tool tip (cutting radius, cutting width, tool-tip position, etc.). You can check in the simulation whether the contour is machined completely or needs to be reworked, whether the contour is damaged by the tool or overlaps are too large. The cutting path graphics is especially useful for recessing or drilling operations as well as for machining slopes where the tool shape has an essential influence on the accuracy of the resulting workpiece.

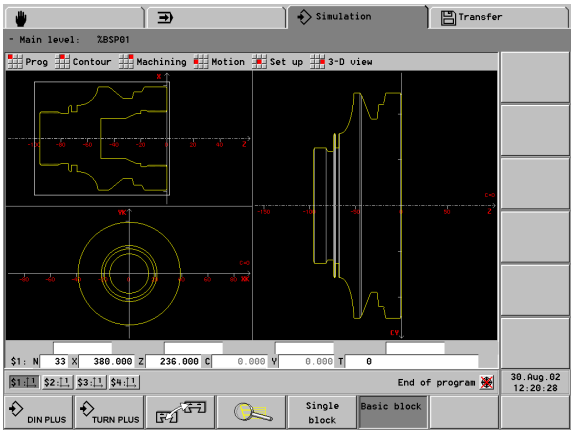


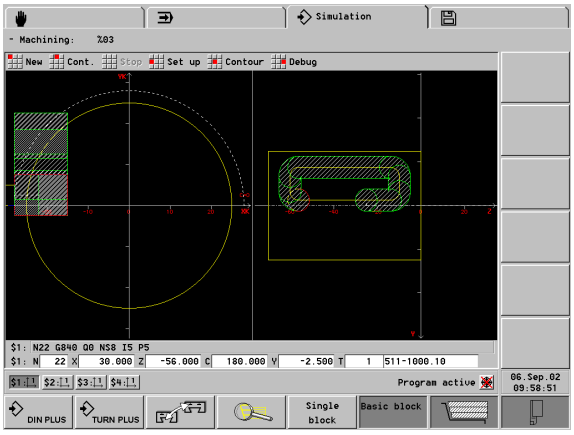


Simulation window

With the simulation windows described in the following you check not only the turning work but also the drilling and milling operations.

- **Turning window:** The turning contour is depicted in the XZ coordinate system.
- **Front window:** The contour and traverse paths are shown in the XY plane, taking the spindle position into account. The spindle position 0° is located on the positive X-axis (designation: XK).
- **Surface window:** The contour display and traverse-path display are oriented to the position on the unrolled lateral surface (designation: CY) and the Z coordinates. Contours on the lateral surface are shown "on the workpiece surface." (Contours of the lateral surface are drawn on the milling floor in the graphic window of the DIN PLUS editor.)
- **Side view (YZ):** The contour and traverse path are shown in the YZ plane. The side view depicts only the Y and Z coordinates—not the spindle position (see figure below).

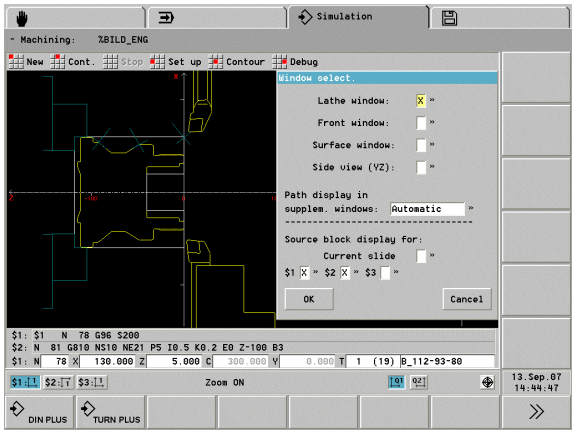




- **Front face and surface windows** operate with a fixed spindle position. Whereas the machine turns the workpiece, the graphic simulation moves the tool.
- The surface window and side view (YZ) are shown alternatively.
- The **surface window** is suited for simulation of drilling, boring and milling operations with the C axis.
- The **side view** is suited for simulation of the Y axis and for operations on tilted planes.

Setting the simulation window

Window selection dialog box:



- Select “Set up > Window”: The CNC PILOT opens the dialog box for the settings listed below.

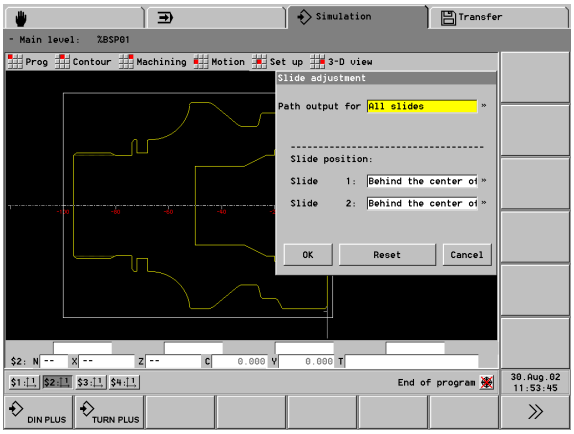
Set:

- The window combination
- The path display in the supplementary windows: The front window, surface window, and side view are considered supplementary windows. The following setting specifies whether the simulation depicts traverse paths in the windows.
  - **Automatic:** The simulation displays traverse paths if the C axis has been oriented or a G17 or G19 has been executed. G18 or a C axis out of orientation stops output of the traverse paths.
  - **Always:** The simulation shows the traverse path in all simulation windows.
- Source block display: The **block display** displays the programmed NC blocks (NC source blocks) of one or more slides. Set:
  - Source block display for the currently specified slides
  - Source block display for the marked slides

Configuring the simulation

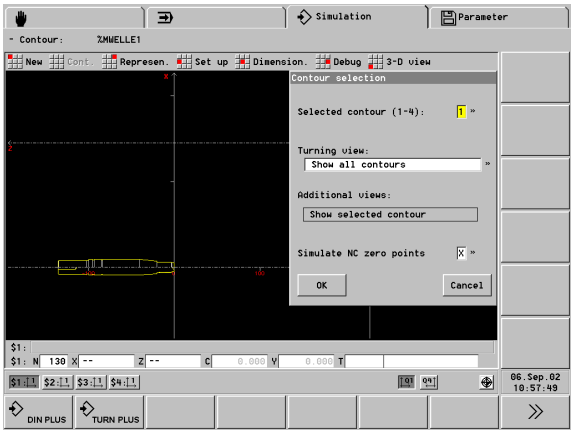
Slide setting:

- Select “Set up > Slide”: The CNC PILOT opens the “slide settings” dialog box for the settings listed below.
  - Path display for all slides: The simulation shows the paths of all slides.
  - Path display for current slide: The simulation shows the paths of the selected slide.
  - Slide position of slide x: The simulations shows the paths of the slide in front of / behind the workpiece.
  - Reset button: The slide position defined in the machine parameters is adopted.



Contour depiction:

- Select “Set up > Contour selection”: The CNC PILOT opens the “contour selection” dialog box for the settings listed below.
  - Selected contour: Contour displayed in the turning view and in the supplementary views.
  - “Show selected contour” turning view: The simulation depicts only the selected contour.
  - “Show all contours” turning view: The simulation depicts all the contours defined in the NC program.
  - Simulating NC zero points: This setting specifies whether zero point shifts are taken into account.



Unrolled surface:

- When the surface window is active, select “Set up > Zero point C”: The CNC PILOT opens the “Zero point” dialog box. Set the angle at which the lateral surface is to be “cut” for unrolling. This angle lies on the Z axis (default setting: C angle = 0°).

Adjusting the section (zoom function)

In the stopped condition, you can magnify/reduce the detail section with the zoom function

Zoom settings by keyboard:



▶ Activate the magnify function The red rectangle indicates the new section.



If several simulation windows are open:

▶ Choose the window.

▶ Adjust the section:

- To magnify, use the PgDn key
- To reduce, use the PgUp key
- To move the frame, use the arrow keys

ESC

▶ Exit the zoom function. The new section is displayed

Zoom settings by touchpad:

▶ Position the cursor to one corner of the section

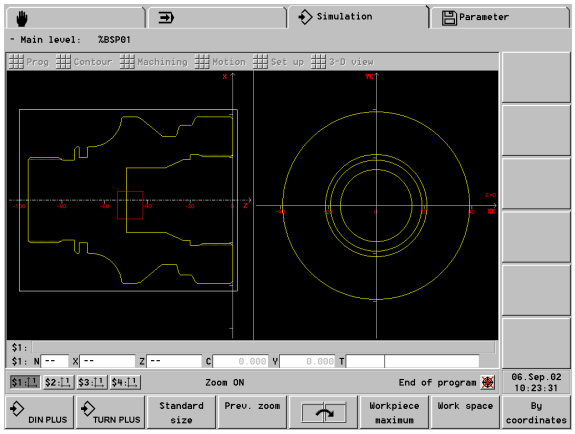
▶ While holding the left mouse key, pull the cursor to the opposite corner of the section

▶ Right mouse key: Return to standard size

ESC

▶ Exit the zoom function. The new section is displayed.

You can make the standard setting by soft key (see table). In the “By coordinates” setting you define the magnification of the simulation window and position of the workpiece datum. The setting applies to the selected slide.



Soft keys for standard settings	
Standard size	Last setting was “workpiece maximum” or “working space”
Prev. zoom	Cancels the last magnification
Workpiece maximum	Depicts workpiece as large as possible
Work space	Depicts the work space including the tool change position
By coordinates	Adjusts the simulation window

Errors and warnings

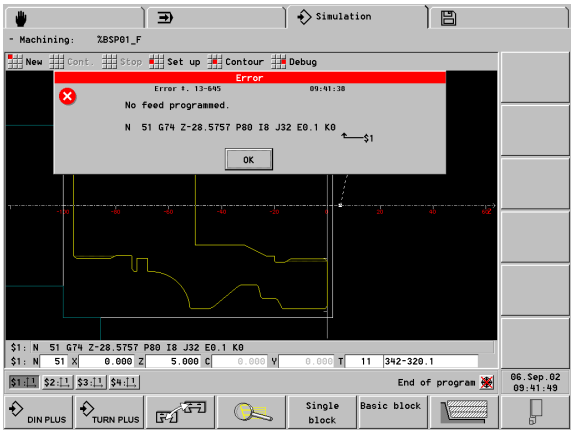
Warnings that occur during the interpretation of an NC program are displayed in the header.

You see these warnings during a simulation stop or after the simulation:

- Select “Set up > Warnings”
- If there are two or more: Press ENTER to switch to the next message

The CNC PILOT deletes a warning after you have confirmed the corresponding message with ENTER. The system stores a maximum of 20 warnings.

If an error occurs during the interpretation of an NC program, the simulation is stopped.



How to activate the simulation function

Loading the NC program:

- Select Prog > Load: The CNC PILOT shows the selection box with all NC main programs.
- Select and load the NC program

Load the NC program from **from DIN PLUS:**

- Select “Prog > From DIN PLUS”

After program changes in the DIN PLUS editor, press “New” to simulate the changed NC program.

Select the type of simulation:

- “Contour” calls the contour simulation
- “Machining” calls the machining simulation
- “Motion” calls the movement simulation
- “3-D view” calls the three-dimensional depiction

Simulation mode

You can set by soft key whether the simulation will be run continuously or blockwise.



- **Single block:** Stop after every NC source block

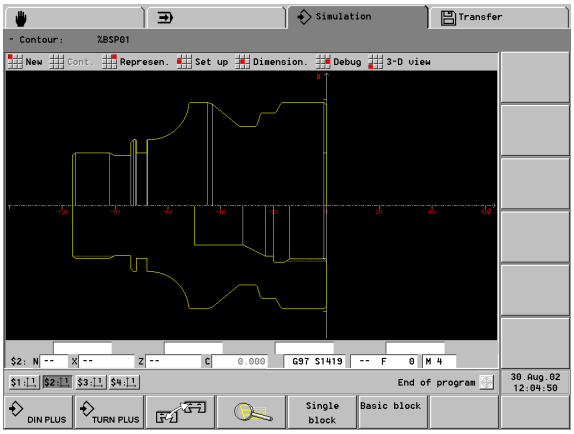


- **Basic block**
  - Contour simulation: Stop after every contour element
  - Machining or motion simulation: Stop after every traverse path
- **Without stop** (single block and basic block soft keys are not pressed): The simulation is conducted without stop.
- Menu item “Stop”: The simulation stops.
- Menu item “Continue”: The simulation resumes.

During a **simulation stop** you can switch the block mode, edit the settings or call the dimensioning function.

5.2 Contour Simulation

Functions of the contour simulation



Contour simulation presupposes that the blank or finished part contour (blank or finished part definition, auxiliary contours) is programmed. If the blank or finished part contours have not yet been completely programmed, they are displayed as completely as possible.

The contour-simulation function allows you to:

- Select between “section” or “view” graphics.
- Check the contour programming through simulation in single blocks.
- Check the parameters of a contour element (element dimensioning).
- Measure each contour point with respect to a reference point (point dimensioning).

Controlling the contour simulation:

- Select “New”: Re-simulates the contour (contour changes are taken into account).
- Select “Continue”: The simulation shows the next NC block or basic block.

Contour depiction:

- Select “Represen.”: The CNC PILOT opens the “Contour representation” dialog box. Set:
  - (Cross-) “Section”
  - (Lateral) “View”
  - Section & View. “View” is used above the center of rotation, “Section” is used below

ESC

- To return to the main menu, press the ESC key



In single or basic block mode, the “section” is shown.

Additional functions:

- Debug menu item: If you use variables for contour description, check them with the debug functions: see “Simulation with starting block” on page 383
- “3-D view” menu item: see “3-D View” on page 382

Contour dimensioning

Position the cursor:

For element or point dimensioning, position the cursor (small red square) as follows:

- Cursor left/right: Changes to the next contour point
- Cursor up/down: Changes the contour (for example, between blank and finished contour)



- Changes to the next simulation window (prerequisite: there are contours on the reference planes).

Element dimensioning

- Select “Dimension. > Element dimension.”
- Position the cursor on a contour element: The simulation shows the data of the marked contour element. The arrow shows the direction of the contour description.

Point dimensioning:

- Select “Dimension. > Point dimension.”

Set reference point:

- Position the cursor to the reference point
- Select “Set reference point”

Measure the contour point:

- Position the cursor to the contour point to be measured: the simulation shows the dimensions of the contour point with respect to the reference point and the selected reference plane (XC, XY, etc.).

Cancel the reference point:

- Select “reference point off”: The simulation deletes the reference point.

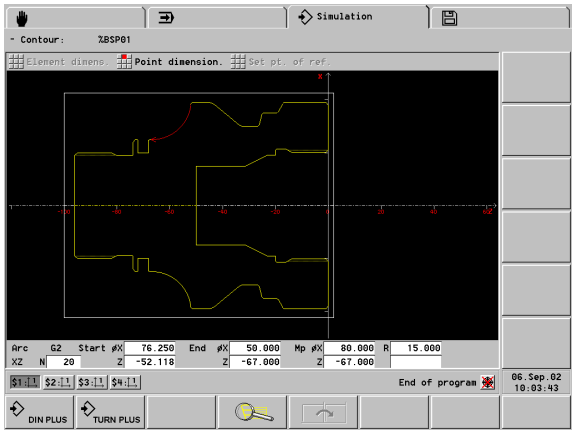
Back to the contour simulation:

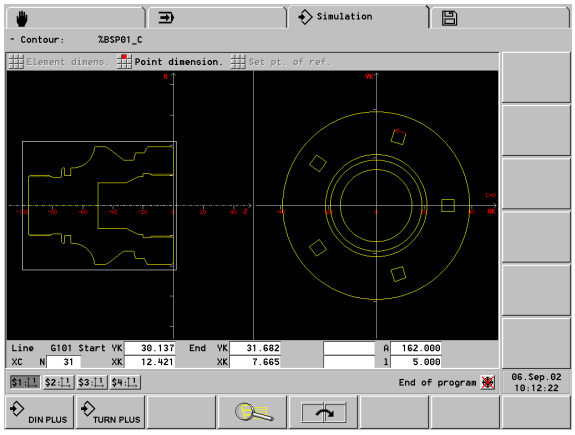


- ESC key



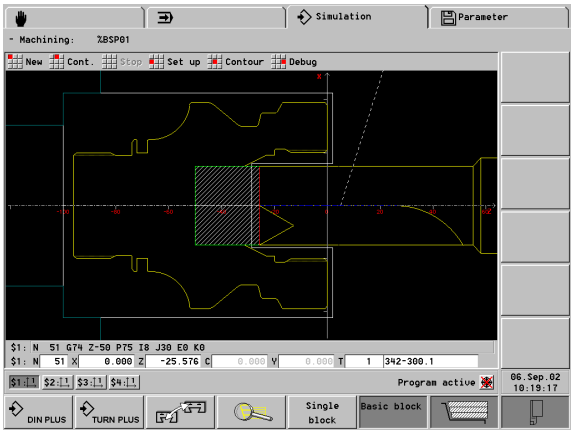
You can also call the **dimensioning functions** in the Machining/Motion mode of simulation (“Dimensioning” menu item).





5.3 Machining Simulation

Checking the workpiece machining



The machining simulation function allows you to:

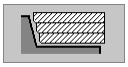
- Check the traverse paths of the tool
- Check the number of cutting passes
- Measure the machining time
- Monitor protection zone and limit switches
- Display and set variables
- Save the machined contour

You can change the speed of the machining simulation with control parameter 27.

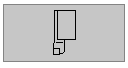
Controlling the contour simulation:

- Select “New”: The CNC PILOT simulates machining again (program changes are taken into account).
- Select “Continue”: The CNC PILOT simulates the next NC block or basic block.
- Select “Stop”: The simulation stops. Thus you can edit the settings or use the contour follow-up function.

Influencing the traverse-path and tool-display functions:



- View of the paths of traverse: **Line** or **(cutting) track**



- Tool depiction: **Point of light** or **tool**

Return to main menu:



- ESC key

Additional functions:

- Menu item “Set up > Warnings”: see “Errors and warnings” on page 372
- Menu item “Set up > Times”: Changes to machining time display (see “Time Calculation, Synchronous Point Analysis” on page 387)
- “Debug” menu item: If you use variables for workpiece description, check them with the debug functions: see “Simulation with starting block” on page 383

Protection zone and limit switch monitoring (machining simulation)

You set the monitoring of protection zones and limit switch as follows:

- "Set up > Protection zone > Monitoring off": The protection zones/ limit switches are not monitored.
- Select "Set up > Protection zone > Monitoring with warning": The CNC PILOT registers protection-zone or limit-switch violations and treats them as warnings. The NC program is simulated up to the end of program.
- Select "Set up > Protection zone > Monitoring with error": A protection zone or limit switch violation results in an immediate error message and cancelation of the simulation.



You define the protection zone dimensions in the setup mode. They are managed in MP 1116, ...

Dynamic limit switch monitoring

Available as of software version 625 952-05.

With dynamic limit switch monitoring, the CNC PILOT monitors the traverse paths of two slides moving on the same guideway for a possible collision. This feature is set by the machine manufacturer.

As the paths of the two slides are not executed in their real order in the simulation, the following simplified test is made:

- At program start and each shared synchronous point, the simulation calculates the slide positions.
- Based on these positions, the simulation checks all traverse paths up to the next shared synchronous point, or up to the end of the program. Within this program section, the paths of the two slides must not overlap.
- If the simulation finds any collision risk, a warning or error is generated.

If necessary, program further synchronous points to separate critical program sections.

Set monitoring as follows:

- "Set up > Dynamic limit switches > Monitoring off": The limit switches are not monitored.
- Select "Set up > Dynamic limit switches> Monitoring with warning": The CNC PILOT registers possible collisions and treats them as warnings. The NC program is simulated up to the end of program.
- Select "Set up > Dynamic limit switches > Monitoring with error": A possible collision results in an immediate error message and cancelation of the simulation.

Contour checking

With the functions of the “Contour” menu you can adapt the contour to the progress of machining, or switch to contour dimensioning or 3-D view.

Contour follow-up:

- Select “Contour > Contour follow-up”: The simulation deletes all previously depicted traverse paths and updates the contour according to the simulated progress of machining. The CNC PILOT takes the blank part as a basis and accounts for each cut.

Displays the contour according to its current state of progress:

- Select “Contour > Contour follow-up”: The simulation updates the contour according to the simulated progress of machining.
- Select “Contour > Dimensioning”: The simulation activates the element and point dimensioning (see “Contour dimensioning” on page 375).

3-D view

- Select “Contour > 3-D view”: The simulation switches to 3-D view (see “3-D View” on page 382).

Saving the generated contour

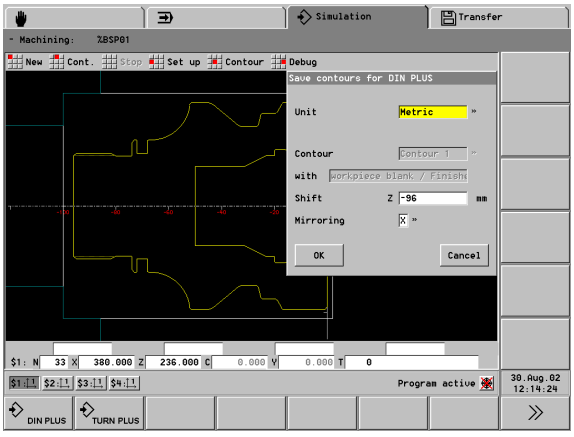
You can save a contour generated in the simulation and read it into DIN PLUS. In DIN PLUS, you insert the workpiece blank into the program and finished part contour that you generated during simulation (block menu “Insert contour”).

Example: You describe the blank form and finished part and simulate machining of the first setup. Then you save the machined contour and use it for the second setup.

During contour generation, the simulation saves:

- BLANK: The simulated contour's state of progress
- FINISHED PART: The programmed finished part

The simulation takes into account a shift of the workpiece zero point and/or a mirror image.

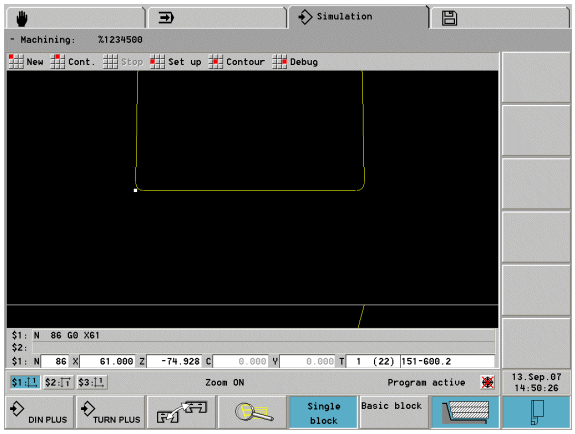


Save the contour:

- Select “Contour > Save contour”: The simulation opens the dialog box “Save contours as NC subprogram.” Input Fields:
  - Unit: Contour description metric or in inches
  - Contour: Selection of the contour (when there are more than one)
  - Shift: Shifting the workpiece zero point
  - Mirroring: Mirror/do not mirror the contours

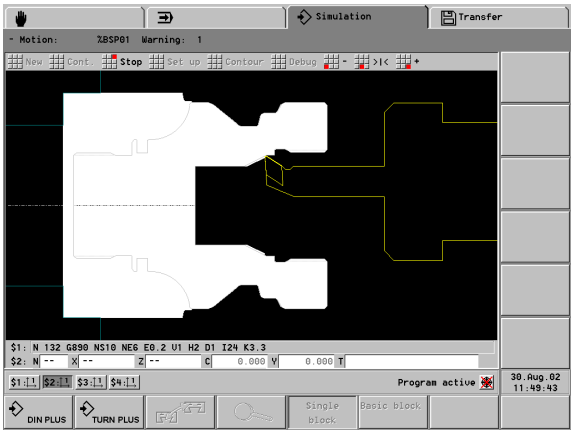
Displaying the tool tip reference point

In the machining simulation, the simulation depicts the tool tip reference point with very high magnification. You can also derive the tool orientation from it.



5.4 Motion Simulation

Real-time simulation



The motion simulation depicts the workpiece blank material as a "filled surface" and "machines" it during simulation by "erasing" the material (erasing graphics). The tools move at the programmed feed rate (program-run graphics).

You can interrupt the motion simulation at any time, even during simulation of an NC block. The display below the simulation window indicates the target position of the current path.

If other simulation windows are active in addition to the lathe window, the machining process is shown as "track-display graphics" in the supplementary windows.

Controlling the contour simulation:

- Select "New": The CNC PILOT simulates machining again (program changes are taken into account).
- Select "Continue": The CNC PILOT simulates the next NC block or basic block.
- Select "Stop": The simulation stops. Thus you can edit the settings or use the contour follow-up function.

Influencing the traversing speed (via menu):

- **"–":** Reduces the traversing speed
- **">|<":** Traversing speed in real time
- **"+":** Accelerates the traversing speed

Return to main menu:

ESC

- ESC key

Additional functions:

- Menu item "Set up > Warnings": see "Errors and warnings" on page 372
- Menu item "Set up > Times": Changes to machining time display (see "Time Calculation, Synchronous Point Analysis" on page 387)
- "Debug" menu item: If you use variables for workpiece description, check them with the debug functions: see "Simulation with starting block" on page 383

Protection zone and limit switch monitoring (motion simulation)

You set the monitoring of protection zones and limit switch as follows:

- "Set up > Protection zone > Monitoring off": The protection zones/ limit switches are not monitored.
- Select "Set up > Protection zone > Monitoring with warning": The CNC PILOT registers protection-zone or limit-switch violations and treats them as warnings. The NC program is simulated up to the end of program.
- Select "Set up > Protection zone > Monitoring with error": A protection zone or limit switch violation results in an immediate error message and cancelation of the simulation.

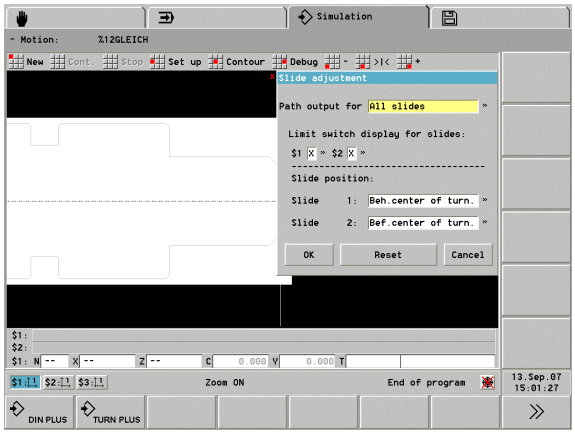
You define the protection zone dimensions in the setup mode. They are managed in MP 1116, ...

Visual limit-switch and protection-zone monitoring

- Select "Set up > Slide": The CNC PILOT opens the "slide settings" dialog box.
- In the "Limit switch display for slides input fields...," specify which limit switches are to be displayed.

Depending on this setting, the motion simulation shows the **Software limit switch** or the protection zone with respect to the tool tip. That simplifies monitoring the traverse paths near the working space limits. The visual monitoring is independent from the protection zone monitoring and limit switch monitoring.

The simulation draws a rectangle resulting from the limit switches and the protection zone. The smaller of the dimensions are depicted. If a limit switch defines a side of the rectangle, the line is shown in red. If the protection zone defines a side, the line is red/white.



The simulation shows the limit switch dimensions with respect to the tool point. This is why the limit switch dimensions are repositioned in a tool change.

Contour checking

With the functions of the "Contour" menu you can switch to contour dimensioning or to the 3-D view.

Displays the contour according to its current state of progress:

- Select "Contour > Dimensioning": The simulation activates the element and point dimensioning (see "Contour dimensioning" on page 375).

3-D view

- Select "Contour > 3-D view": The simulation switches to 3-D view (see "3-D View" on page 382).

5.5 3-D View

Influencing the 3-D view

In the 3-D view the CNC PILOT shows the workpiece in its simulated condition. If you call the 3-D view from the main menu or from the contour simulation, the finished part is depicted.



The 3-D view includes contours machined by turning, but not for C or Y or B operations.

Calling 3-D view:

▶ Select “3-D view” or “Contour > 3-D view”

Standard 3-D view

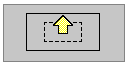
▶ Depiction as “solid-body model” in the standard view (not rotated, not enlarged/reduced)

Grating

▶ Depiction as wire model

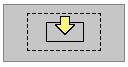
To rotate the workpiece:

▶ Press the cursor keys, plus key or minus key



To magnify the graphic:

▶ Press the soft key or the PgDn key



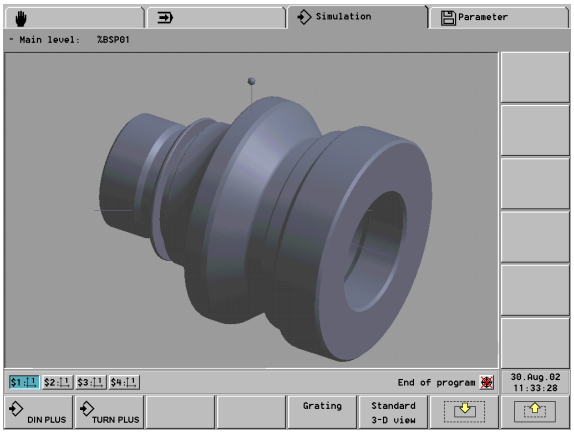
To reduce the displayed graphic:

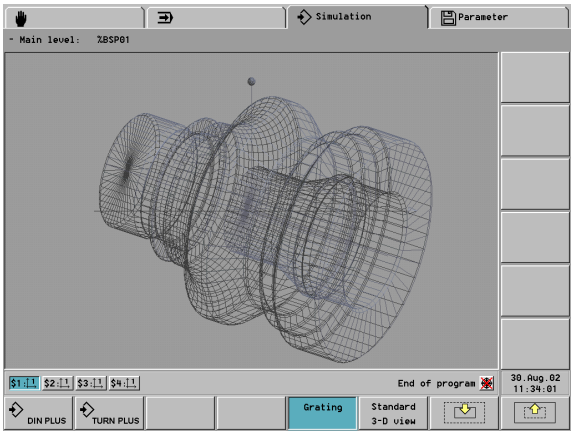
▶ Press the soft key or the PgUp key

ESC

To exit the 3-D view:

▶ ESC key





5.6 Debugging Functions

Simulation with starting block

If a start block is defined, the simulation converts the NC program without displaying the traverse to the starting block.

Setting the start block:

- Select “Debug > Set starting block”: The simulation opens the “Set start block” dialog box.
- Enter the block number
- Select “New”: The CNC PILOT simulates the NC program up to the starting block and stops.
- Select “Continue”: The CNC PILOT resumes the simulation.

Delete the starting block:

- Select “Debug > Delete starting block“: The starting block is removed.

Checking the start block:

- Select “Debug > View start block”: The simulation displays the starting block.

Displaying variables

Permanent variables display: Instead of the NC source block, the simulation shows four “selected variables” below the simulation window.

Select the variables:

- Select Debug > Display variables > Set display: The simulation opens the “Selection of display” dialog box.
- Setting the variable type and variable number

Activate the variable display:

- Use “Debug > Variables/Source block” to set the variables display

Select the variables:

- Select “Debug > Display variables > Reset display”: The simulation removes the selected variables.

Display # variables in the dialog box:

- Select “Debug > Variables display > All # variables.” The simulation displays the variables in the “# variables“ dialog box.

Navigating within the dialog box:

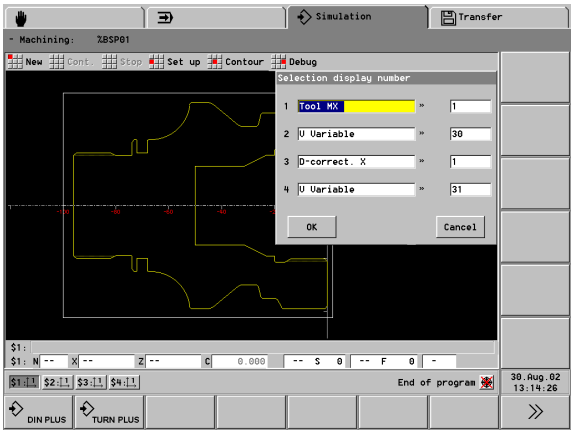
- Cursor up/down or PgUp/PgDn

Display V variables in the dialog box:

- Select “Debug > Display variables > All V variables”: The simulation opens the “V display” dialog box for the following input:
  - Variable type
  - Number of the first variable to be displayed
- The simulation displays the variables in the “V variables“ dialog box.

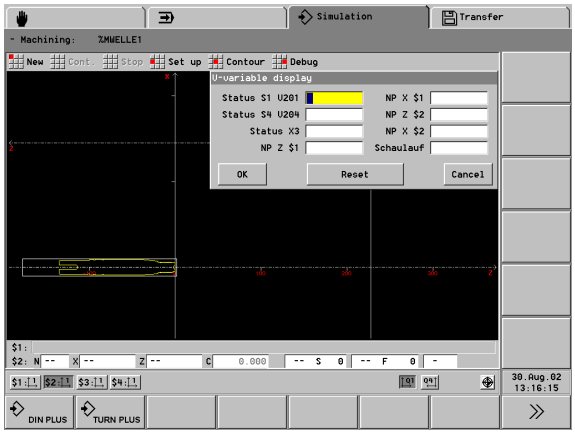
Navigating within the dialog box:

- Cursor up/down or PgUp/PgDn



Editing variables

For complex NC programs with branches, variable calculations, events, etc., you simulate all inputs and events to test all program branches.



Editing variable values:

- Select “Debug > Edit variables > Edit all V variables”: The simulation opens the “Edit V variables” dialog box:
- The “Edit V variables” dialog box:
  - Setting the variable type and variable number
  - Preset the value or the event
  - Define the status (see following list)

Meaning of the status (“Edit V variables” dialog box):

- **Undefined:** No value/event is assigned to variable. That is the same as the condition after the NC program start. During simulation of an NC block with this variable, the simulation prompts you to enter the value/the event.
- **Defined:** During simulation of an NC block with this variable, the entered value/event is adopted.
- **Request:** Each time an NC block containing the variable is simulated, the system requests the variable value/event.

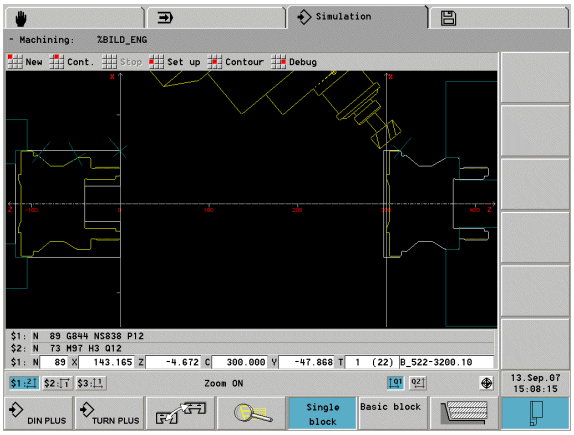
Deleting variable values:

- Select “Debug > Edit variables > Delete all xx variables”: The simulation deletes the variables or events. The xx stands for:
  - V variables
  - D compensation variables
  - Event variables
  - Machine dimension variables
  - Tool variables

5.7 Checking Multi-channel Programs

The simulation provides the following checking capabilities for NC programs with two or more slides:

- Analysis of all contours (workpieces) defined in the NC program
- Checking the movements of all slides
- Detecting collision risks through true-to-scale depiction of the workpieces, tools and chucking equipment
- Time calculation, separate for each slide and tool use (machining times and non-cutting time)
- Examining the temporal progress of machining with the synchronous point analysis



The **start block display** shows the slides that you have selected (see “Displays” on page 364).

The **coordinate system** is shown for the workpiece you have selected (see “Displays” on page 364).

Debug functions enable you to view and set variables. In this way you can simulate all branches of the multi-channel program (see “Debugging Functions” on page 383).

5.8 Time Calculation, Synchronous Point Analysis

Time Calculation

During machining or motion simulation, the CNC PILOT calculates the productive and non-productive times. The display appears in the “time calculation” table The table shows the simulation of the machining times, idle times and total times (green: machining times; yellow: idle times). Each line represents the use of a new tool (for each tool call with T).

If there are more table entries than fit on a screen page, you can call further time data with the arrow keys and PgUp/PgDn.



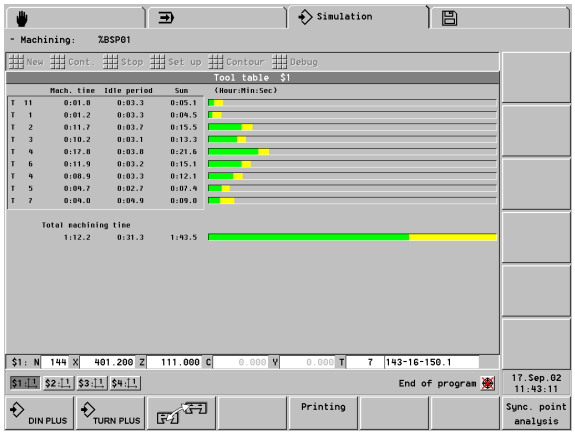
In control parameters 20 and 21 you can define which control parameters will be used for calculating the idle times.

You can call the time calculation at any time that the simulation is stopped:

▶ Select “Set up > Times”

Exit the time calculation:

ESC ▶ ESC key



Soft keys	
	Switch to the next slide
Printing	Output the time calculation (see “General control parameters” on page 577).

Synchronous point analysis

The synchronous point analysis shows the chronological history of machining and the interdependency of the slides. This helps you to organize and optimize a multi-channel program. Information in the synchronous point analysis:

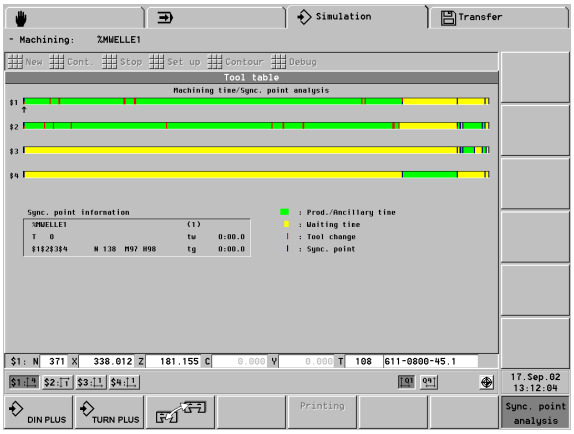
- Machining times/idle times
- Waiting times
- Tool change
- Synchronous points

Synchronous point information:

- The NC block relevant for the selected synchronous point
- tw: Waiting time at this synchronous point
- tg: Calculated execution time as of program start

Evaluating the synchronous point analysis: Place the cursor (arrow beneath the bar diagram) on the event to be analyzed in order to obtain the following synchronous point information:

- NC program/subprogram
- Type of event (tool change or synchronous point)
- Slide involved
- Active tool
- NC block number
- tw: Waiting time at this synchronous point
- tg: Calculated execution time as of program start



Calling the synchronous point analysis

▶ Select “Set up > Times”

Sync. point analysis

▶ Press the soft key

Selecting the next/previous synchronous point:

▶ Right/left arrow



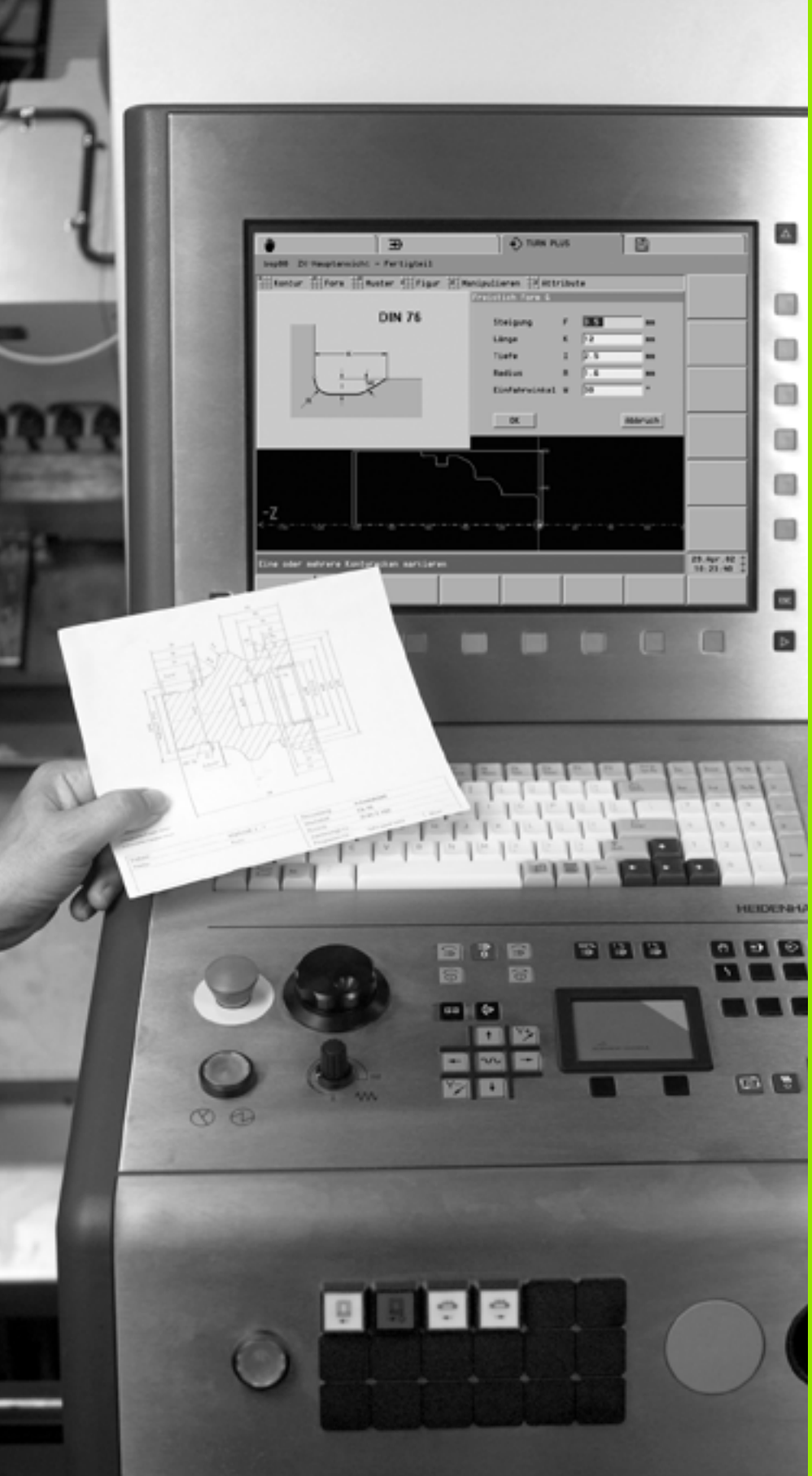
Change slides:
▶ Press the soft key or PgUp/PgDn

Sync. point analysis

Return to time calculation:
▶ Press the soft key again

ESC

Return to simulation:
▶ ESC key



6

TURN PLUS

6.1 TURN PLUS Mode of Operation

TURN PLUS enables you to describe the workpiece blank and finished part in a graphic-interactive environment. Then you can have the working plan generated fully automatically or generate it yourself interactively. The result is a commented and structured NC program.

TURN PLUS features:

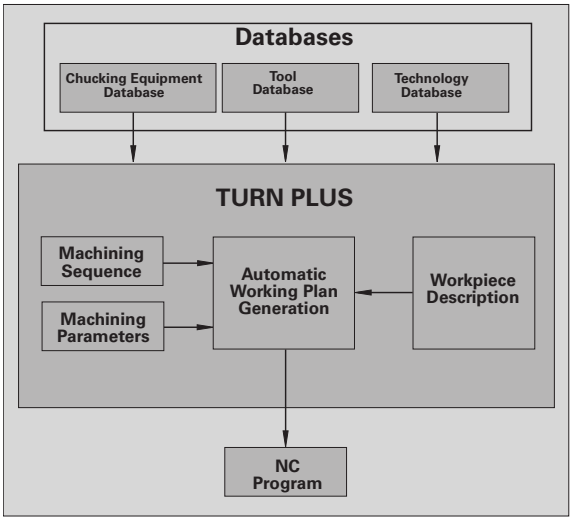
- Graphic-interactive contour definition
- Set-up functions (for chucking the workpiece)
- Interactive working plan generation (IWG)
- Automatic working plan generation (AWG)

For

- Turning operations
- Drilling and milling with the C axis
- Drilling and milling with the Y axis
- Full-surface machining

TURN PLUS concept

The workpiece description is the basis for working plan generation. The strategy for generating the working plan is specified in the **machining sequence.** The **machining parameters** define details of machining. This allows you to adapt TURN PLUS to your individual needs.



TURN PLUS generates the working plan, which takes technology attributes such as oversizes, tolerances, peak-to-valley height, etc. into account. Each entry and working step are displayed and can be corrected immediately.

On the basis of **workpiece blank regeneration**, TURN PLUS optimizes the paths for approach and avoids idle cuts or collisions between workpiece and cutting edge.

For tool change, TURN PLUS provides the following strategies:

- Automatic selection from the tool database
- Use of the current turret assignment
- TURN PLUS turret assignment

When clamping the workpiece, TURN PLUS determines the cutting limitations and the zero point shift for the NC program.

The technology database provides the cutting data for the AWG/IWG.

You can use part of the functions and continue working in DIN PLUS (example: Define a contour using TURN PLUS and program the machining operation in DIN PLUS). Alternately, you can optimize the DIN PLUS program generated by TURN PLUS.



The working plan generation uses the tool/chucking equipment and the technological information contained in the respective database. Therefore, make sure that the database contains a correct description of the operating resources.

TURN PLUS files

TURN PLUS has separate directories for:

- Complete programs (blank and finished part definition as well as working plan generation)
- Workpiece descriptions (blank and finished parts)
- Descriptions of workpiece blanks
- Descriptions of finished parts
- Independent contours
- TURN PLUS turret assignment

You can use this structure for your requirements, for example, for generating different working plans using the same workpiece definition.

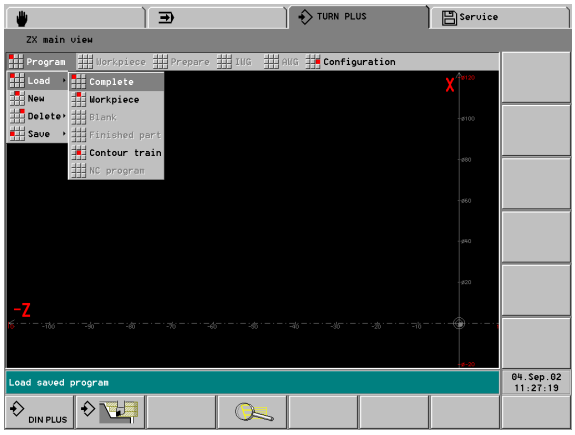
TURN PLUS program management

Creating a new program:

- Select “Program > New”. TURN PLUS opens the “New program” dialog box.
- Enter the program name and select the material.
- Press the “Program head” button: TURN PLUS changes to the program head editor.
- Edit the program head and close the dialog box. TURN PLUS creates the new program.
- Define the blank and the finished part.
- Generate the working plan.

Loading a program:

- Select “Program > Load > Complete (or Workpiece, ..)”. TURN PLUS displays the files.
- Select and load the file. TURN PLUS shows the selected contour(s) and presents it/them for editing.



To generate a DIN PLUS program:

- Select "Program > Save > NC program". TURN PLUS displays the existing DIN PLUS programs and presents the currently active program for saving.
- Check/correct the file name.
- TURN PLUS generates the DIN PLUS program when saving the file.

Saving TURN PLUS programs:

- Select "Program > Save > Complete (or Workpiece, ..)". TURN PLUS displays the existing files of the directory and presents the currently active program for saving.
- Check/correct the file name and save the file.



When you select "Save > Complete", TURN PLUS saves the blank and finished part definition as well as working plan, and generates the DIN PLUS program.

Deleting TURN PLUS programs:

- Select "Program > Delete > Complete (or Workpiece, ..)". TURN PLUS displays the files.
- Select the file and delete it

Operating notes

TURN PLUS uses a multi-level menu structure. The "ESC" key switches one level back.

This description includes operation by mouse, soft keys and touch pad. However, you can still operate the CNC PILOT, as in earlier versions, without soft keys or touch pad.

The "status line" (above the soft key row) informs you how to proceed.

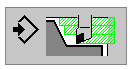
If **more than one window** (view) is displayed on the screen, the "active window" is identified by a green frame.

- PgUp/PgDn switches between the windows.
- The period key "." displays the active window as a full screen. Pressing the period key "." again switches back to multiple windows.



X values can be entered as diameter or radius values, depending on the configuration.

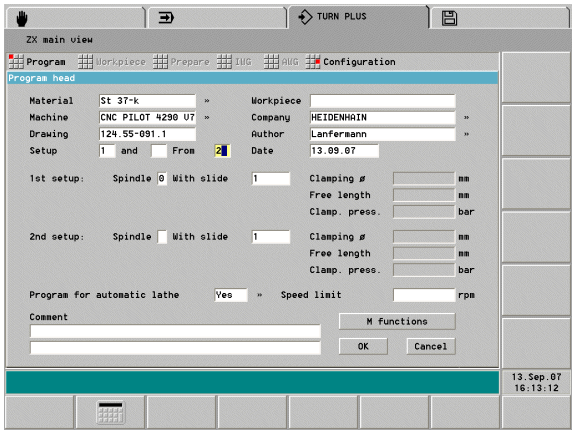
For further notes on the configuration, see "Configuring TURN PLUS" on page 549.

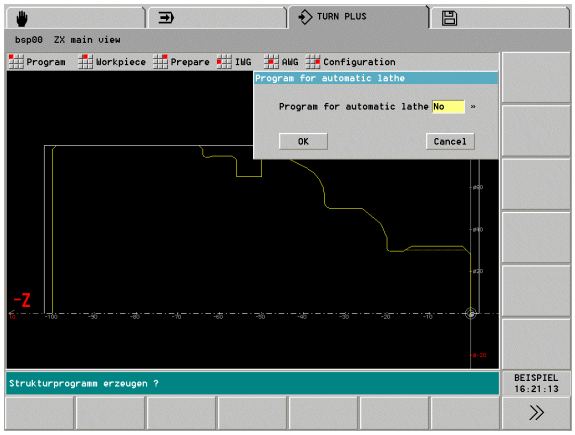
Soft keys	
	Change to the DIN PLUS mode
	Change to the simulation mode

6.2 Program Head

The PROGRAM HEAD comprises:

- **Material:** For determining the cutting values.
- **Assignment of spindle to slide for 1st setup**
- **Assignment of spindle to slide for 2nd setup:** For full-surface machining, enter the spindle and the slide for machining the setup. If more than one slide is used, enter the slide numbers one after the other (for example: “12” = $1 and $2).
- Speed limit (SMAX is defined in “Machining parameter 2 – Global technology parameters”):
  - No input: SMAX is the speed limit
  - Input < SMAX: The entered value is the speed limit
  - Input > SMAX: SMAX is the speed limit
- **“M functions” button:** You can define up to five M functions, which will be taken into account by TURN PLUS when generating the NC program:
  - At the start of machining
  - After a tool change (T command)
  - At the end of machining
- **“Program for automatic lathe” field:** When you select “Yes”, TURN PLUS generates the NC program as a program for an automatic lathe (prerequisite: the workpiece is full-surface machined on a machine with opposing spindle). For each machining operation, an internal subprogram is generated. The main program contains the general commands and the subprogram calls.





You can also change the setting of the “Program for automatic lathe” field via a dialog box. To open the dialog box, select “Workpiece > Program for automatic lathe”.

When you select “Prepare”, TURN PLUS determines the following data of the program head (see “Chucking a workpiece at the spindle” on page 483):

- Clamping diameter
- Free length
- Clamping pressure

The other fields contain **organizational information** and **set-up information**, which do not influence the machining process.

Information contained in the program head is preceded by # in the DIN program.

Generating programs for automatic lathes

Use the following settings to generate a DIN PLUS program for automatic lathes by using TURN PLUS:

▶ Set the "Program for automatic lathe" program head entry to YES

Prerequisite: The templates "turnvor1.bev" to "turnvor5.bev" are provided in the "/ep90/ncps" directory. The templates are prepared by the machine tool builder and used for generating DIN PLUS programs.

As of software version 625 952-05: In **Template management** (machining parameter 23) you set whether constants should be output during generation of a program for automatic lathes.

- 0: Without output of constants
- 1: With output of constants

The templates provide the following functions:

- "turnvor1.bev" sets the display of structure variables in the program head
  - #ANZEIGE V200 "Status S0 V200"
  - #ANZEIGE V203 "Status S3 V203"
- "turnvor2.bev" defines the start of MACHINING at the position marked with [[?-TURNPLUS-?]] and adds TURN PLUS program start information.
- "turnvor3.bev" defines the work block. The following interface is used:
  - [[la; s=workblock no. (n)]
  - [lb; s=slide number]
  - [lc; s=spindle number]
  - [ld; s=1 if subpgm; e=S]
  - [le; s=1, if altern.SP; e=S]
  - [i; s=workblock hdr?; e=S]
  - [j; s=if spindle state =]
  - [k; s=subprogram]
  - [o; s=comment SP$1]
  - [p; s=then spi. state =]
  - [r; s=spindle at top?]
  - [s; s=TURN PLUS comment]
  - [u; s=T number]
  - [w; s=tool ID number]]
- "turnvor4.bev" defines the rechucking operation. The expert program call is inserted at the position of the [[?-TURNPLUS-?]] placeholder. The block number for the return jump to the start of program in M99 is given in #__la.
- "turnvor5.bev" defines those constants in the CONST section that are specific to the machine tool builder.

As of software version 625 952-05: In the templates, you can use the following constant designators, which will be replaced by data from TURN PLUS: :

?-TP_MINFD-?	Minimum inside diameter of the finished part
?-TP_MAXFD-?	Maximum outside diameter of the finished part
?-TP_FINL-?	Length of the finished part
?-TP_MINFZ-?	Minimum finished part coordinate, 1st setup
?-TP_MAXFZ-?	Maximum finished part coordinate, 1st setup
?-TP_MINRD-?	Minimum outside diameter of workpiece blank at the end of the 1st setup
?-TP_MAXRD-?	Maximum inside diameter of workpiece blank at the end of the 1st setup
?-TP_RAWL-?	Length of workpiece blank at the end of the 1st setup
?-TP_MINRZ-?	Minimum coordinate of workpiece blank at the end of the 1st setup
?-TP_MAXRZ-?	Maximum coordinate of workpiece blank at the end of the 1st setup
?-TP_CLAMD1-?	Clamping diameter, main spindle
?-TP_INCLA1-?	Clamping length, main spindle
?-TP_OUTCLA1-?	Free length, main spindle
?-TP_CLAMD2-?	Clamping diameter, opposing spindle
?-TP_INCLA2-?	Clamping length, opposing spindle
?-TP_OUTCLA2-?	Free length, opposing spindle
?-TP_MAXG026-?	Maximum speed, spindle 0
?-TP_MAXG126-?	Maximum speed, spindle 1
?-TP_MAXG226-?	Maximum speed, spindle 2
?-TP_MAXG326-?	Maximum speed, spindle 3
?-TP_ZPZ1-?	Datum shift, main spindle
?-TP_ZPZ2-?	Datum shift, opposing spindle
?-TP_ZPOZ-?	Datum offset

Die **Bearbeitungszyklen** werden für jeden Bearbeitungsblock in ein internes Unterprogramm geschrieben. The following syntax is used for generating the subprogram names:

- **$Snn** where:
  - $ = slide number
  - S = spindle number (0 to 3)
  - nn = Operation number

6.3 Workpiece Description

You program a contour by entering individual contour elements one after the other in the correct sequence. You define the contour elements using absolute, incremental, Cartesian or polar coordinates. You can usually program a contour with the dimensions given in the workpiece drawing.

TURN PLUS automatically calculates all missing coordinates, points of intersection, center points, etc. that can be derived mathematically. If the entered data permit several possible solutions, you can inspect the individual solutions and select the proposal that matches the drawing.

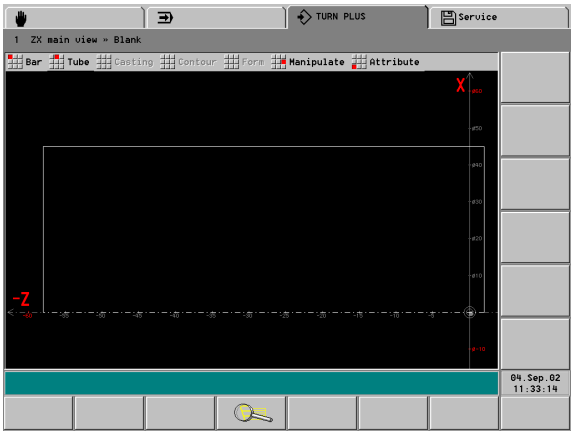
You can import the following contours if they are available in **DXF format** (see "Importing of DXF Contours" on page 458):

- Workpiece blanks
- Finished parts
- Contour trains
- Milling contours

Entering the workpiece blank contour

To describe the workpiece blank, proceed as follows:

- Standard forms (bars, pipes): Definition with workpiece blank macros
- Complex workpiece blanks: Description as for finished part
- Cast or forged blanks are generated from the finished part and the oversize



Additional information:

- see "Contours of Workpiece Blanks" on page 402
- see "Attributes for workpiece blanks" on page 470

Entering the workpiece blank contour

Select "Workpiece > Blank > Bar" (".. > Tube" or ".. > Casting").

Enter the dimensions of the workpiece blank or oversize.

The CNC PILOT displays the workpiece blank.

ESC

Press the ESC key to return to the main menu.

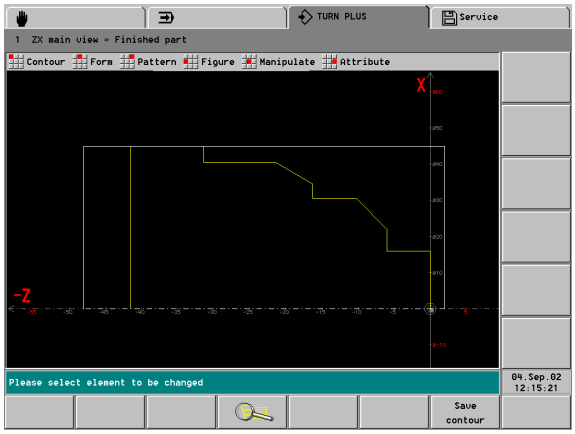
Entering the finished part contour

The finished part contour includes:

- Turning contour, consisting of
  - Basic contour
  - Form elements (chamfers, rounding arcs, undercuts, recesses, threads, centric holes)
- C-Axis Contours
- Y-axis contours

The turning contour must be a **closed** contour.

Describe first the basic contour and then superimpose the form elements.



Additional information:

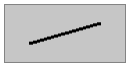
- see “Notes on defining contours” on page 404
- see “Help Functions” on page 447
- see “Assigning Attributes” on page 470

Entering the basic contour

Select “Workpiece > Finished part > Contour.”

Specify the starting point of contour.

Entering the basic contour element for element (see also the illustration of the menu structure):



For linear elements:

- Call the lines menu.
- Select the direction from the menu symbols.
- Describe the line segment.

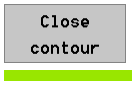


For circular arcs:

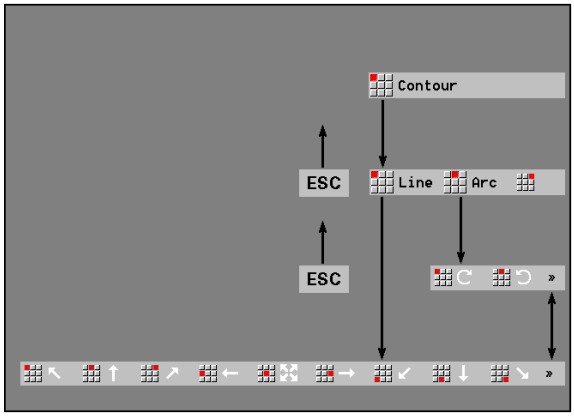
- Call the arcs menu.
- Select the direction from the menu symbols.
- Describe the arc.



Press the ESC key to go back one menu level.

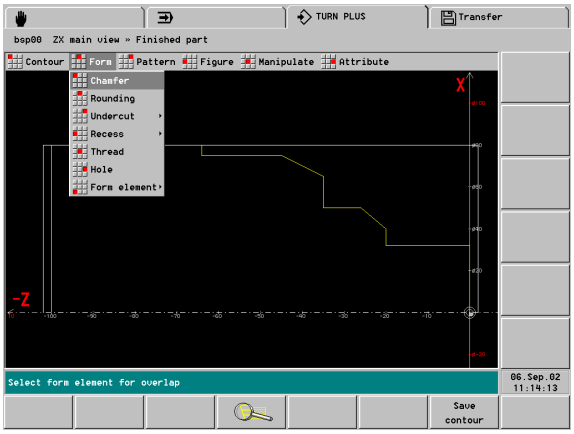


If required: Close the contour.



Superimposing form elements

Form elements are **superimposed** on the basic contour. They remain independent elements that can be edited or deleted. If required, TURN PLUS generates a special machining cycle for the form elements.



The selection considers the type of form element:

- **Chamfer**: Outside corners
- **Rounding**: Outside and inside corners
- **Undercut**: Inside corners with paraxial straight lines located at right angles
- **Recess**: Straight lines
- **Thread**: Straight lines
- **(Centric) Hole**: Center line on front or rear face

Define chamfers, rounding arcs, undercuts, etc. as **form elements**. Then the working plan generation can take into account special operations on these form elements.

Further information: see “Form Elements” on page 408

Superimposing form elements

Select “Workpiece > Finished part > Form > xx”. (xx: type of form element).

Select the position (see “Selections” on page 448).

Enter the parameters of the form element.

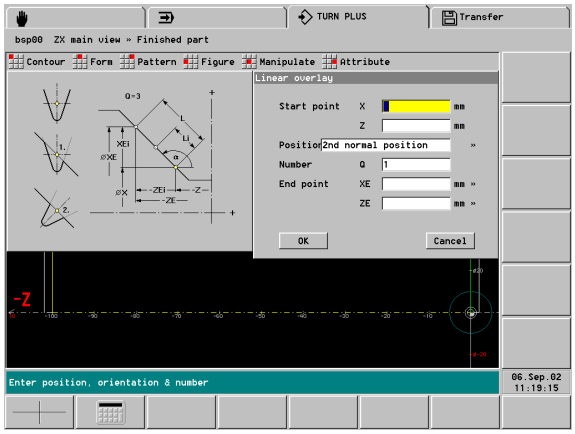
TURN PLUS integrates the form element.

Integrating overlay elements

You describe contour trains in the same way as a finished part contour and superimpose them. Alternatively, you can use the following standard overlay elements (see “Overlay Elements” on page 418):

- Circular arc
- Wedge
- Pontoon

These elements superimpose existing linear or circular supporting contour elements. Integrated overlay elements are part of the contour.



Integrating a contour train:

Select “Program > Load > Contour train”. Select and load the file.

ESC Press the ESC key to return to the main menu.

Select “Workpiece > Finished part > Form > Contour overlap > Contour.”

Integrating a standard overlay element:

Select “Workpiece > Finished part > Form > Contour overlap > xx” (xx: circular arc, wedge or pontoon).

TURN PLUS opens the corresponding dialog box.

Describe the overlay element.

Select a supporting contour element. TURN PLUS opens the “Linear/ Circular overlay” dialog box.

Define the overlay. If more than one solution is displayed, select the desired solution.

TURN PLUS displays the overlay. You can accept (OK) or reject it (Cancel).

TURN PLUS integrates the overlay contours into the existing contour.

Entering contours machined with the C axis

You define standard forms with figures; regular linear or circular figures or holes in patterns. To define complex contours, use the **basic elements** "line" and "arc."

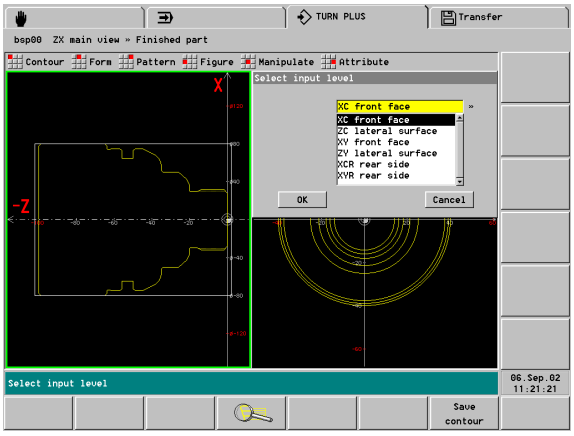
- Patterns
  - Linear hole patterns (drilling patterns)
  - Circular hole patterns (drilling patterns)
  - Linear figure patterns (milling contours)
  - Circular figure patterns (milling contours)
- Figures
  - Circle (full circle)
  - Rectangle
  - Polygon
  - Slot, linear
  - Circular slot
- You position patterns and figures on the
  - Front face (C-axis machining)
  - Lateral surface (C-axis machining)
  - Rear side (C-axis machining)



Define the complete turning contour before defining the contours for C-axis machining.

Selecting the input level

To define a C-axis contour, start by selecting the **"input level"** (front face, lateral surface, rear face). Proceed as follows:



1. Select new window (it is not yet displayed):

- Select the "Turning contour" window.
- Select the pattern/figure from the "Pattern" or "Figure" drop-down menu. TURN PLUS opens the "Select input level" dialog box.
- Select the input level: TURN PLUS opens the corresponding window.

2. Select window (it is already displayed, but not activated):

- Select the window with the PgUp/PgDn keys.

Defining a C-axis contour

Select “Workpiece > Finished part > Pattern > xx”. (xx: pattern type or single hole).

Select “Workpiece > Finished part > Figure > xx”. (xx: figure type or free contour).

Select the front/rear face or lateral surface.

Specify the reference plane (plane on the front/rear face or lateral surface) and enter the reference dimension/diameter. TURN PLUS opens the corresponding dialog box.

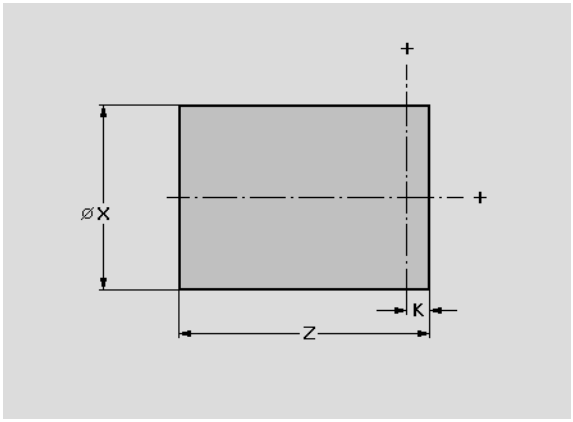
Define the pattern, figure, single hole or contour.

Further information: see “C-Axis Contours” on page 421

6.4 Contours of Workpiece Blanks

Bar

This function defines the contour of a cylinder (chuck or bar).



Parameters

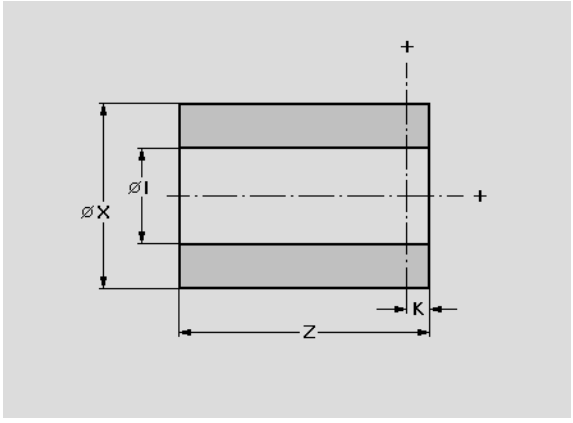
X
- Diameter
- Diameter of circumference of a polygonal blank

Z Blank length, including transverse oversize

K Transverse oversize

Tube

This function defines the contour of a hollow cylinder.



Parameters

X
- Diameter
- Diameter of circumference of a polygonal blank

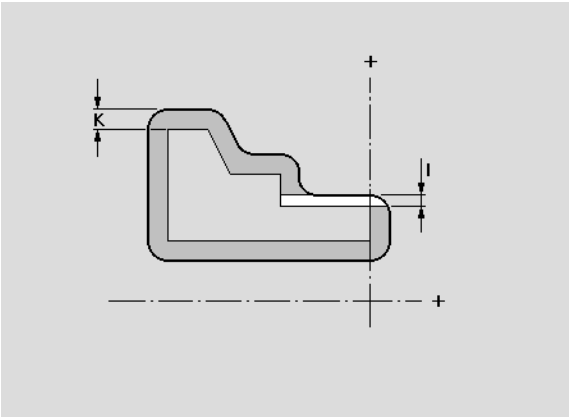
I Inside diameter

Z Blank length, including transverse oversize

K Transverse oversize

Cast blank (or forged blank)

This function generates the workpiece blank from an existing finished part.



Parameters

Surface

- Cast blank
- Forged blank

With bore

- Yes
- No

K Equidistant oversize for the complete part

I Single oversize (for individual elements or contour sections)

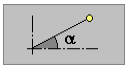
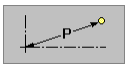
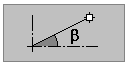
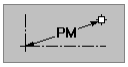
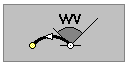
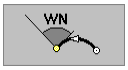
First enter the “single oversize” and then select the contour element/the contour section.

6.5 Contours of Finished Parts

Notes on defining contours

Parameters that TURN PLUS knows are not requested. The input boxes are locked. Example: On horizontal or vertical lines, only one of the coordinates changes and the angle is defined by the direction of the element.

You set the type of dimensioning by soft key.

Soft keys	
	Polar dimensions of the end point: Angle a
	Polar dimension of the end point: Radius
	Polar dimension of the center: Angle b
	Polar dimension of the center: Radius
	Angle to the predecessor element
	Angle to the successor element

Starting point of contour

This function defines the starting point.

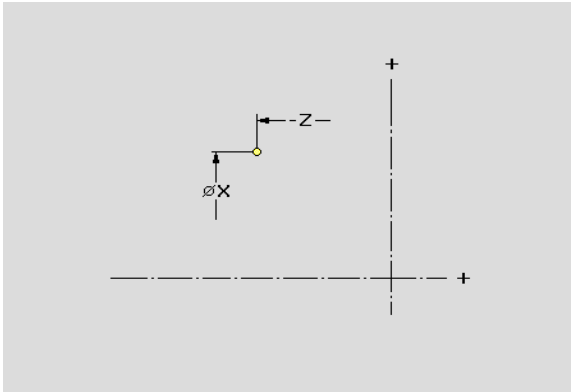
Parameters

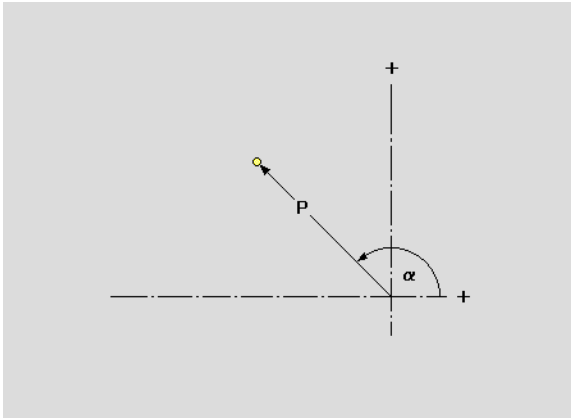
X Starting point of contour

Z Starting point of contour

P Starting point of the contour in polar coordinates

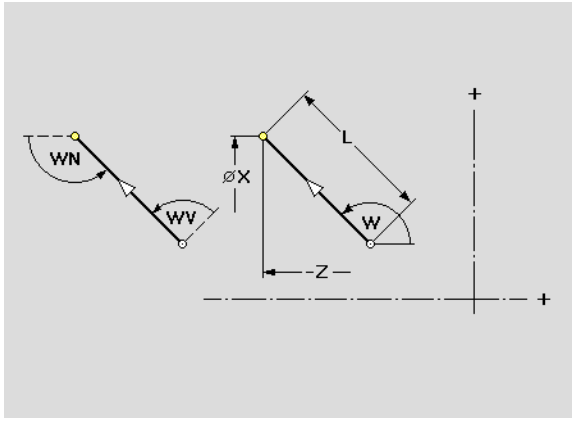
a Starting point of the contour in polar coordinates (reference angle: positive Z axis)

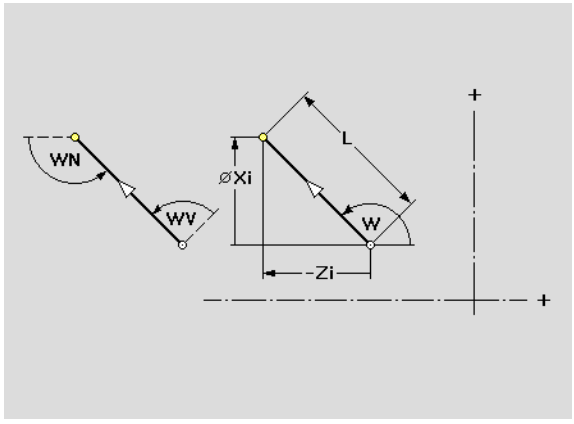


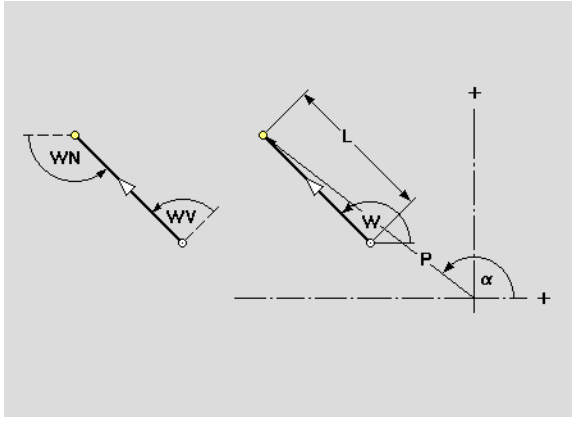


Linear elements

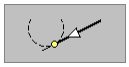
This function defines a line segment.



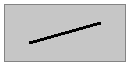




Parameters

X	End point in Cartesian coordinates
Z	End point in Cartesian coordinates
Xi	Distance from starting point to end point
Zi	Distance from starting point to end point
a	End point in polar coordinates (reference: positive Z axis)
P	End point in polar coordinates
W	Angle of the line (for reference see illustration)
WV	Counterclockwise angle to the preceding element. If the preceding element is an arc: Angle to the tangent.
WN	Counterclockwise angle to the following element. If the following element is an arc: Angle to the tangent.
L	Length of element
	Tangential/nontangential: Specify the transition to the next contour element

Defining a linear element:

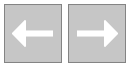


Call the lines menu.

Select the line direction.



Vertical line



Horizontal line



Line at angle



Line at angle

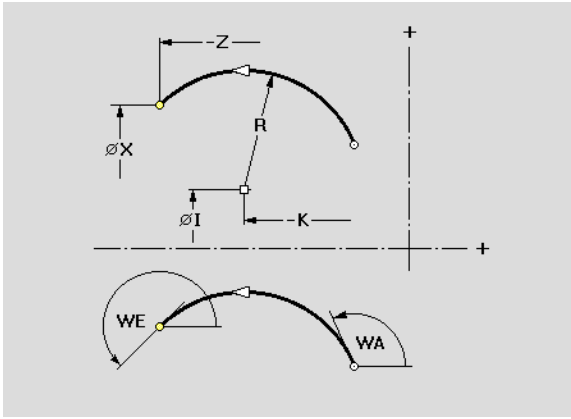


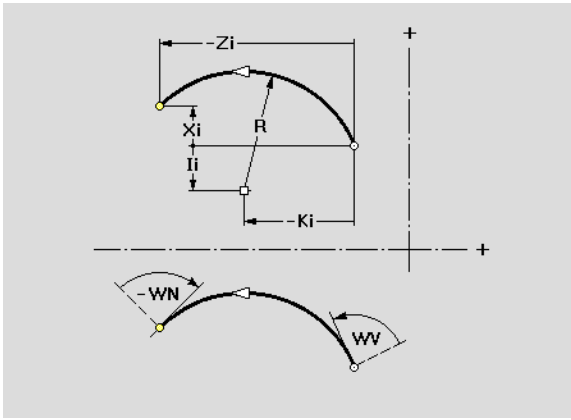
Line in any direction

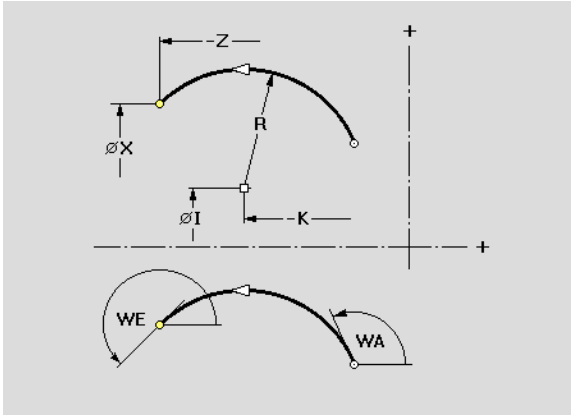
Enter the line dimensions and define the transition to the next element.

Circular element

This function defines a circular arc.







Parameters

End point of arc

X	End point in Cartesian coordinates
Z	End point in Cartesian coordinates
Xi	Distance from starting point to end point
Zi	Distance from starting point to end point
a	End point in polar coordinates (reference: positive Z axis)
P	End point in polar coordinates
ai	End point polar, incremental (for reference of angle ai see illustration)
Pi	End point polar, incremental (linear distance from starting point to end point)

Center of arc

I	Center point
K	Center
Ii	Distance from starting point to center point
Ki	Distance from starting point to center point
b	Center point in polar coordinates (reference: positive Z axis)
PM	Center point in polar coordinates
bi	Center point polar, incremental (angle between an imaginary line intersecting the starting point and parallel to the Z axis, and another line from the starting point to the center point)
PMi	Center point polar, incremental (PMi: linear distance from starting point to center point)

Other parameters

R	Arc radius
	Tangential/nontangential: Specify the transition to the next contour element
WA	Angle between positive Z axis and tangent in starting point of arc
WE	Angle between positive Z-axis and tangent in end point of arc
WV	Counterclockwise angle between preceding element and tangent in starting point of arc. If the preceding element is an arc: Angle to the tangent.
WN	Counterclockwise angle between tangent in arc end point and following element. If the following element is an arc: Angle to the tangent.

Defining a circular element:

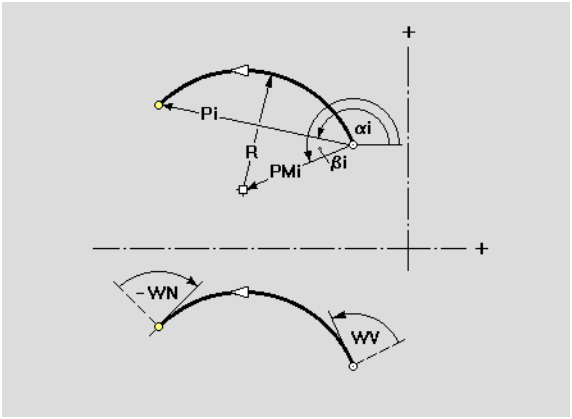


Call the arcs menu.



Select the direction of rotation.

Enter the arc dimensions and define the transition to the next element.



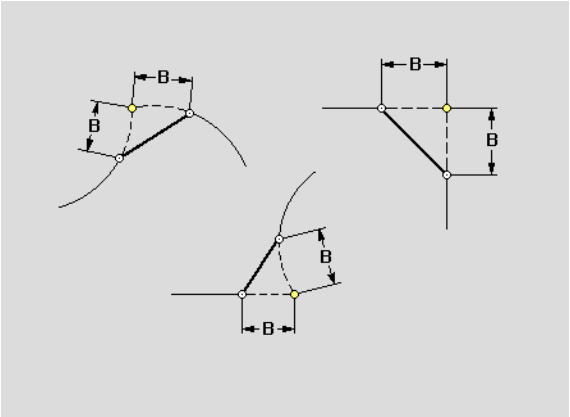
6.6 Form Elements

Chamfer

This form element defines a chamfer.

Parameters

B Chamfer width

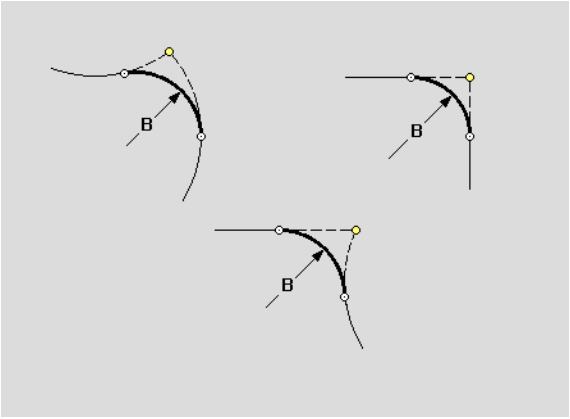


Rounding

This form element defines a rounding.

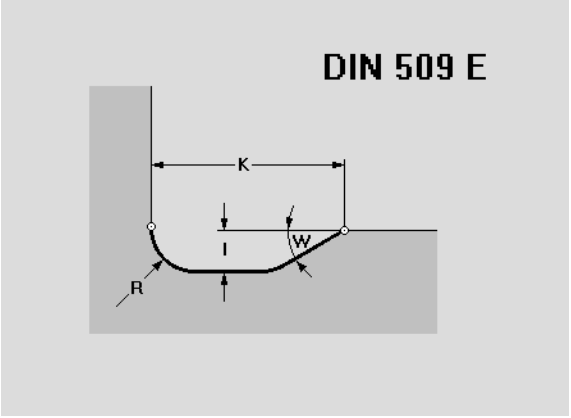
Parameters

B Rounding radius



Undercut type E

This form element defines an undercut type E. TURN PLUS suggests the parameters based on the diameter (see "Undercut DIN 509 E, Parameters" on page 686).



Parameters

K Undercut length

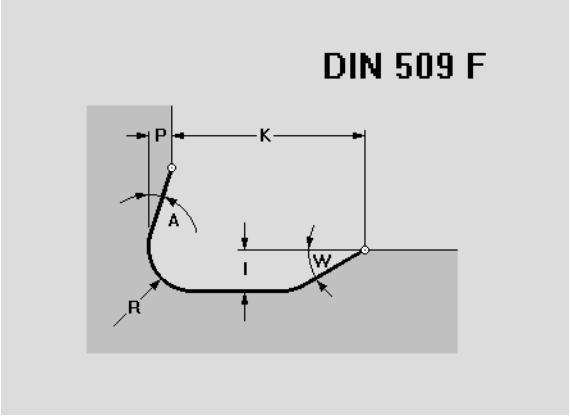
I Undercut depth (radius)

R Undercut radius in both corners of the undercut

W Approach angle (undercut angle)

Undercut type F

This form element defines an undercut type F. TURN PLUS suggests the parameters based on the diameter (see "Undercut DIN 509 F, Parameters" on page 686).



Parameters

K Undercut length

I Undercut depth (radius)

R Undercut radius in both corners of the undercut

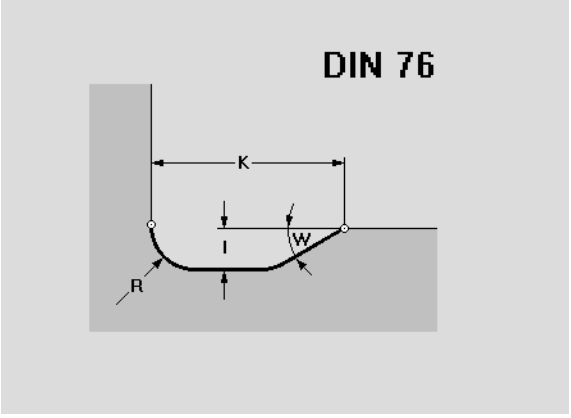
W Approach angle (undercut angle)

A Runout angle (transverse angle)

Undercut type G

This form element defines an undercut type G. TURN PLUS suggests the parameters. You can overwrite the values, if required. The proposed values are based on the metric ISO thread (DIN 13) that is found from the diameter.

- Parameters: see "Undercut DIN 76, Parameters" on page 684
- Determining the thread pitch: see "Thread pitch" on page 688



Parameters

F Thread pitch

K Undercut length (undercut width)

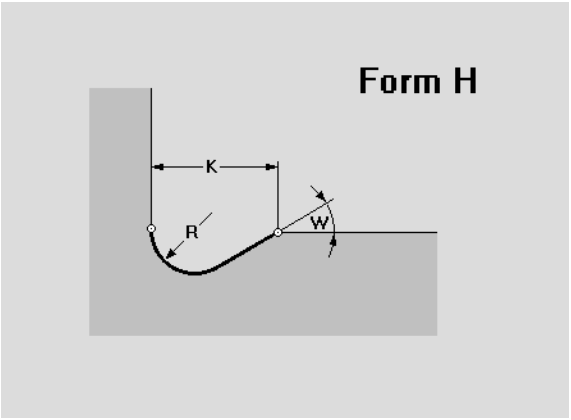
I Undercut depth (radius)

R Undercut radius in both corners of the undercut (default: R=0.6*I)

W Approach angle (undercut angle)

Undercut type H

This form element defines an undercut type H.



Parameters

K Undercut length

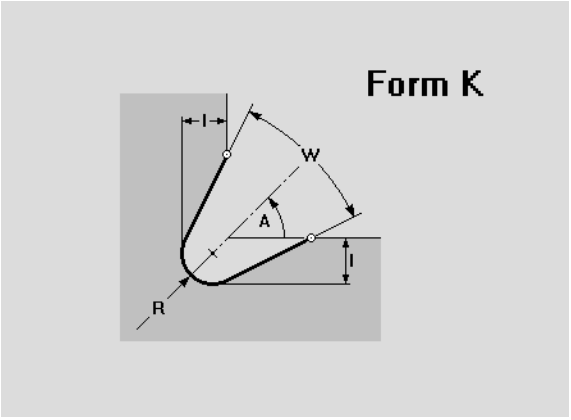
I Undercut depth (radius)

R Undercut radius

W Approach angle

Undercut type K

This form element defines an undercut type K.



Parameters

I Undercut depth

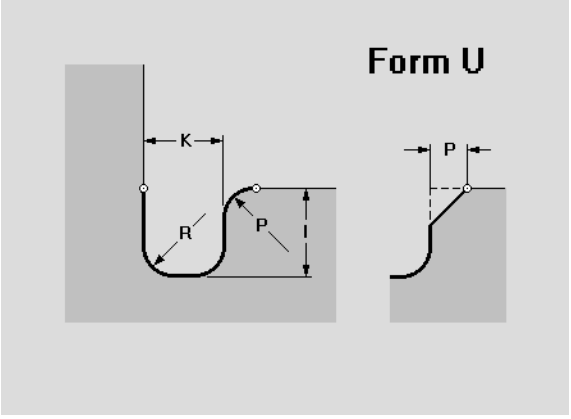
R Undercut radius

W Angular length

A Approach angle, angle to longitudinal axis (default: 45°)

Undercut type U

This form element defines an undercut type U.



Parameters

K Undercut length (undercut width)

I Undercut depth (radius)

R Undercut radius in both corners of the recess (default: 0)

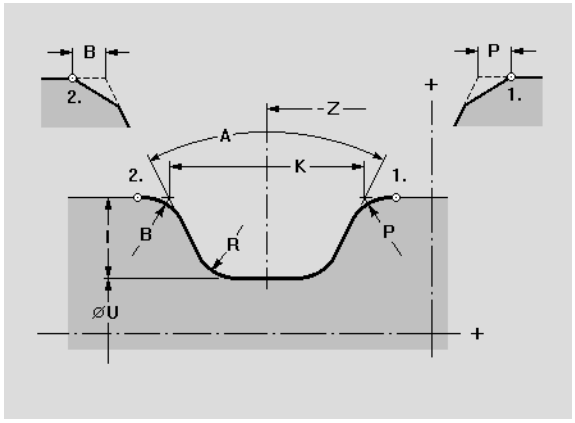
Corner:

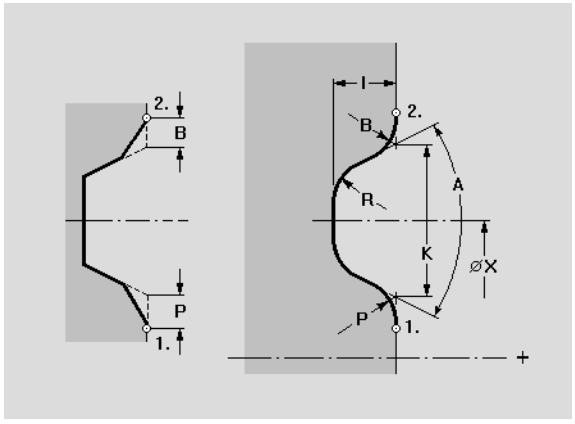
- No: No chamfer/rounding
- Chamfer: Chamfered edge
- Rounding: Rounded corner

P Width of chamfer / radius of rounding

Recess general

This form element defines an axial or radial recess on a linear reference element. The recess is assigned to the selected reference element.





Parameters

X Reference point

Z Reference point

K Recess width without chamfer/rounding arc

I Recess depth

U Diameter of recess base (only for axial recess)

A Recess angle, angle between the recess edges (0° <= A < 180°)

1. Corner:

- No: No chamfer/rounding
- Chamfer: Chamfered edge
- Rounding: Rounded corner

P Width of chamfer / radius of rounding (1st corner)

2. Corner:

- No: No chamfer/rounding
- Chamfer: Chamfered edge
- Rounding: Rounded corner

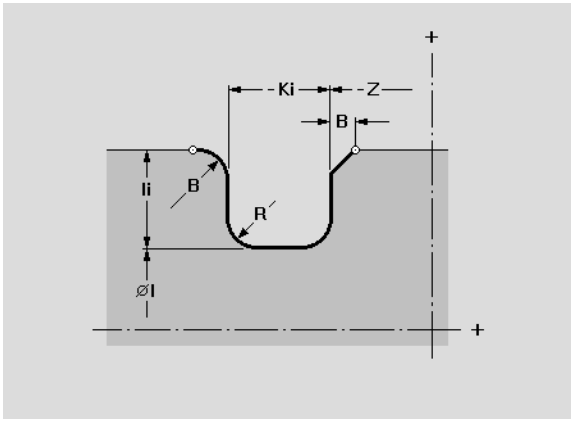
B Width of chamfer / radius of rounding (2nd corner)

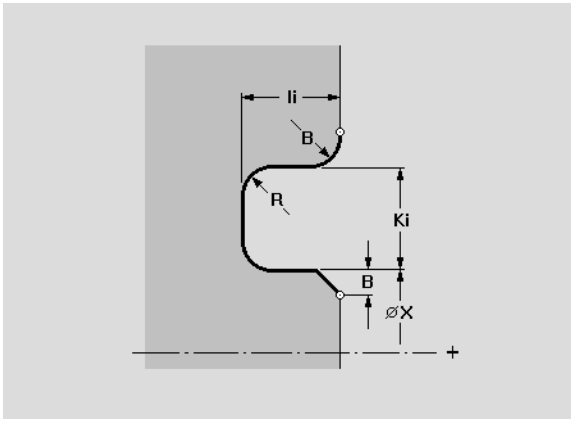
R Radius at bottom (inside radius in both recess corners)

The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.

Recess type D (sealing ring)

This form element defines an axial or radial recess on the outside or inside of the contour. The recess is assigned to the previously selected reference element.





Parameters

X Starting point for radial recess

Z Starting point for axial recess

I Diameter of recess base (only for axial recess)

li
- Axial recess: Recess depth
- Radial recess: Recess width (pay attention to sign!)

Ki
- Axial recess: Recess width (pay attention to sign!)
- Radial recess: Recess depth

Corners:
- No: No chamfer/rounding
- Chamfer: Chamfered edge
- Rounding: Rounded corner

B Width of chamfer or radius of rounding at both sides of the recess

R Radius at bottom, inside radius in both recess corners

Relief turn (type FD)

This form element defines an axial or radial relief turn on a linear reference element. The relief turn is assigned to the previously selected reference element.

Parameters

X Reference point

Z Reference point

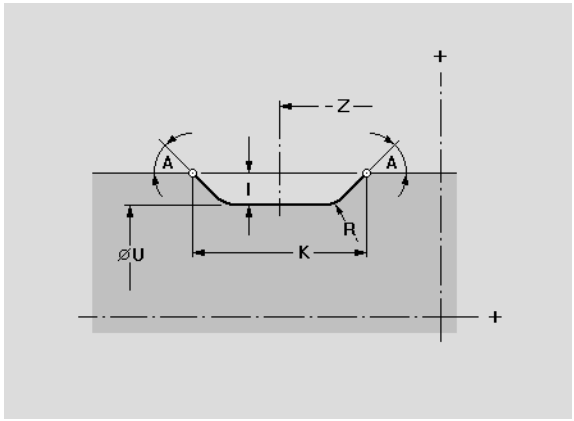
K Recess width

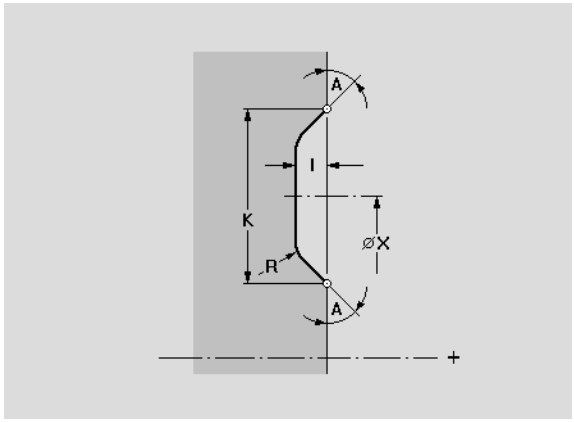
I Recess depth

U Diameter of recess base (only for axial recess)

A Recess angle (0° < A <= 90°)

R Inside radius in both corners of the recess





The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.

Recess type S (guarding ring)

This form element defines an axial recess on the outside or inside of the contour. The recess is assigned to the previously selected reference element.

Parameters

Z Starting point of recess

Ki Recess width (pay attention to sign!)

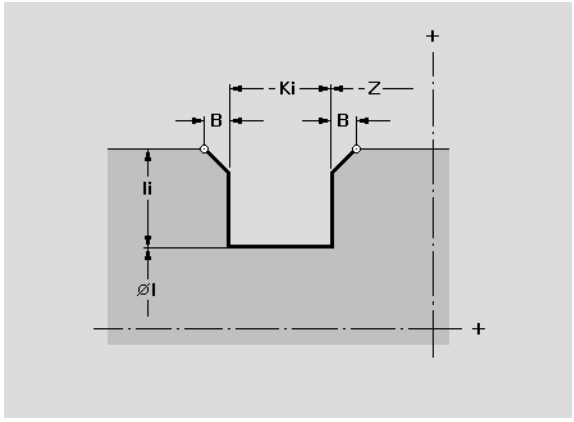
I Diameter/radius of recess base

Ii Recess depth

Chamfer at both sides of the recess

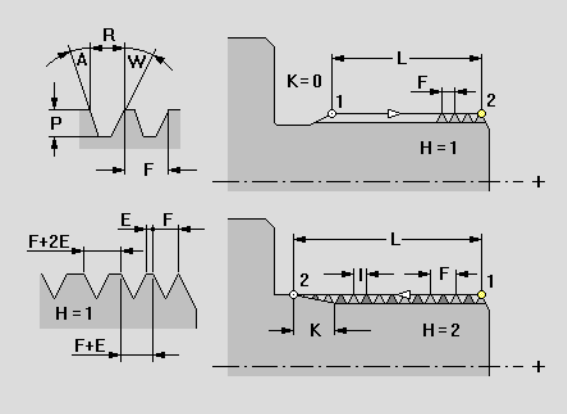
- No: No chamfer
- Chamfer: Chamfered edge

B Width of chamfer



Thread

This function defines the different types of thread.



Parameters

Q Types of threads:

- Metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
- Metric ISO thread (DIN 13 Part 1, Series 1)
- Metric ISO tapered thread (DIN 158)
- Metric ISO tapered fine-pitch thread (DIN 158)
- Metric ISO trapezoid thread (DIN 103 Part 2, Series1)
- Flat metric trapezoid thread (DIN 380 Part 2, Series 1)
- Metric buttress thread (DIN 513 Part 2, Series 1)
- Cylindrical round thread (DIN 405 Part 1, Series 1)
- Cylindrical Whitworth thread (DIN 11)
- Tapered Whitworth thread (DIN 2999)
- Whitworth pipe thread (DIN 259)
- Nonstandard thread
- UNC US coarse thread
- UNF US fine-pitch thread
- UNEF US extra-fine-pitch thread
- NPT US taper pipe thread
- NPTF US taper dryseal pipe thread
- NPSC US cylindrical pipe thread with lubricant
- NPFS US cylindrical pipe thread without lubricant

V Direction:

- Right-hand thread
- Left-hand thread

D Select reference point (see soft key table):

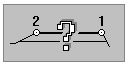
- 1: Beginning of thread at starting point of element
- 2: Beginning of thread at end point of element

F Thread pitch or threads/inch (see soft key table)

- Thread pitch
- Threads/inch

E Variable pitch, increases/reduces the pitch per revolution by E (default: 0)

"Thread" soft keys

	Select reference point
inch	"Thread pitch" or "Threads/inch"

Parameters

L Length of the thread, including runout length

K Runout length (for threads without undercut)

I Division for determining the threads/unit

H Number of thread turns (default: 1)

A Thread angle left, with nonstandard thread

W Thread angle right, with nonstandard thread

P Thread depth, with nonstandard thread

R Thread width, with nonstandard thread



- "F" needs to be defined for metric fine-pitch threads, tapered threads and tapered fine-pitch threads, trapezoid and flat trapezoid threads as well as nonstandard threads. The parameter may be omitted for the other types of threads. The thread pitch is then calculated from the diameter.
- Enter either "I" or "H". The rule is: Thread pitch / graduation = number of starts.
- You can assign further **attributes** to the thread (see "Machining attributes – Threading" on page 474).
- Use the "nonstandard thread" if you want to use individual parameters.



Danger of collision!

The thread is generated to the length of the reference element. For the machining of threads without an undercut, it is necessary to program the "runout length K" so that the overrun can be executed by the CNC PILOT without danger of collision.

(Centric) Hole

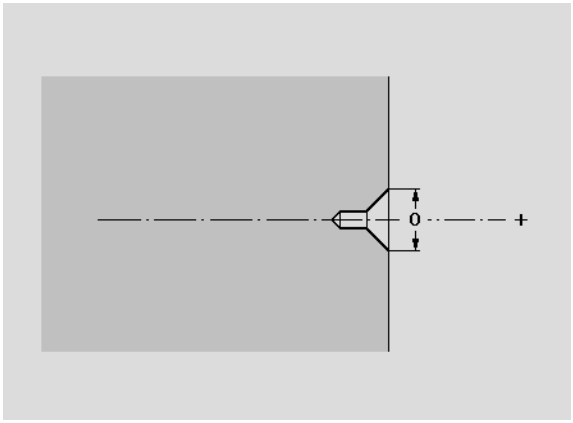
This form element defines a single hole at the turning center (front or rear face). The hole may contain the following elements:

- Centering
- Core hole
- Countersinking
- Thread

Centering

Centering parameters

O Centering diameter



Core hole

Core hole parameters

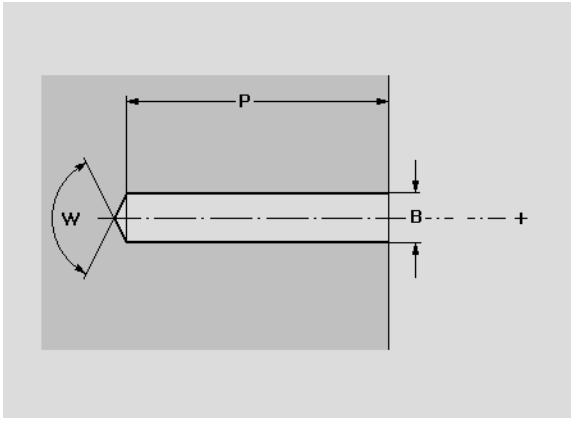
B Hole diameter

P Depth of hole (excluding point)

W Point angle

- W=0°: the AWG generates a "feed rate reduction (V=1)" for the drilling cycle
- W>0°: point angle

Fit: H6 to H13 or "without fit" (see "Drilling" on page 556)



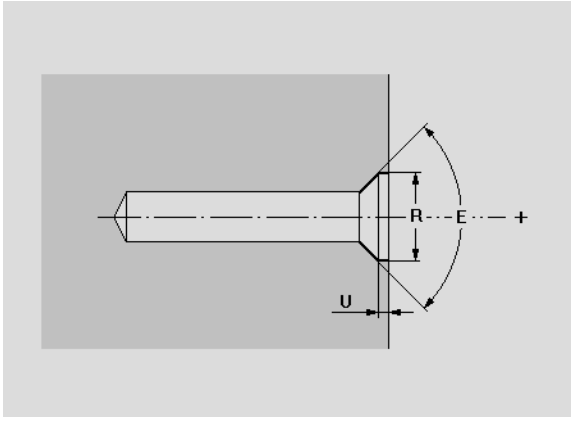
Countersinking

Countersinking parameters

R Sinking diameter

U Sinking depth

E Sinking angle



Tapping

Thread parameters

I Nominal diameter

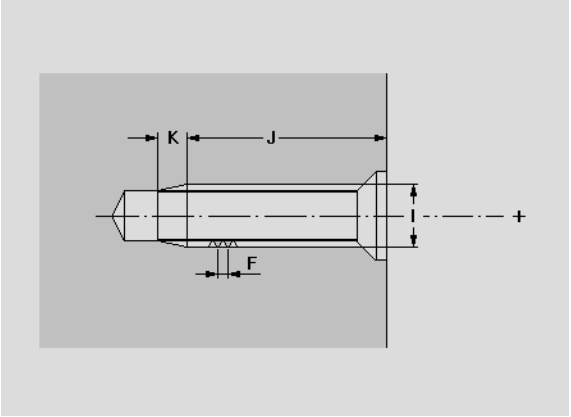
J Thread depth

K K Thread runout length

F Thread pitch

Type of thread:

- Right-hand thread
- Left-hand thread



6.7 Overlay Elements

Select the standard overlay elements arc, wedge or pontoon, define the element and superimpose it after the definition. When you are superimposing a contour train, TURN PLUS uses the last active contour train or the last overlay element defined (see “Integrating overlay elements” on page 399).

Depending on the form of the supporting contour elements, there is a

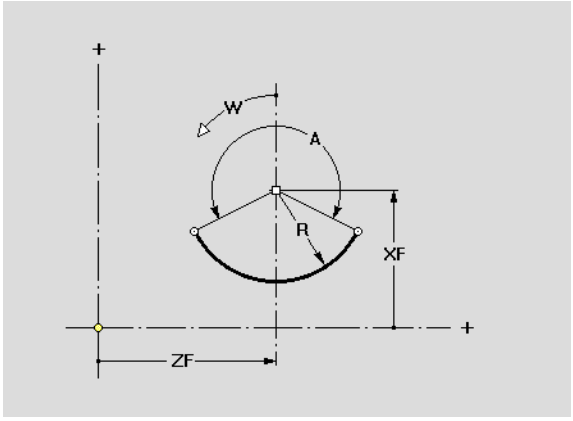
- Linear superimposition (“linear overlay”)
- Circular superimposition (“circular overlay”)



The overlay positions can deviate from the supporting contour element.

Circular arc

Circle center as reference.

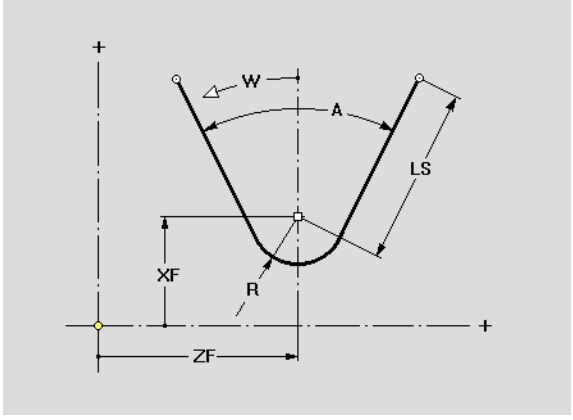


Parameters

XF	Reference point shift
ZF	Reference point shift
R	Radius of the arc
A	Angular length
W	Angle of rotation: The overlay contour is rotated by the “angle of rotation.”

Wedge/rounded wedge

Reference point: Wedge tip/center of rounding arc

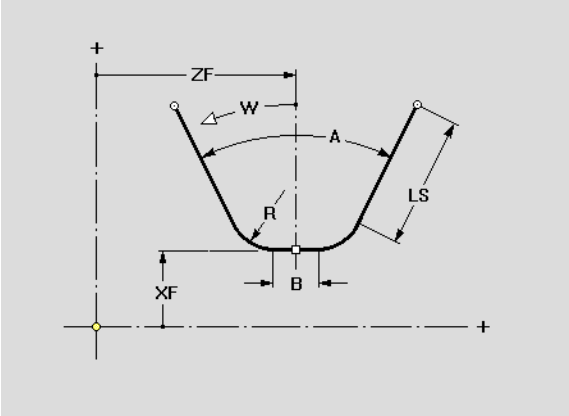


Parameters

XF	Reference point shift
ZF	Reference point shift
R	■ R>0: Rounding radius ■ R=0: No rounding
A	Angular length
LS	Length of wedge sides (projecting element parts are cut at the “points of overlay”)
W	Angle of rotation: The overlay contour is rotated by the “angle of rotation.”

Pontoon

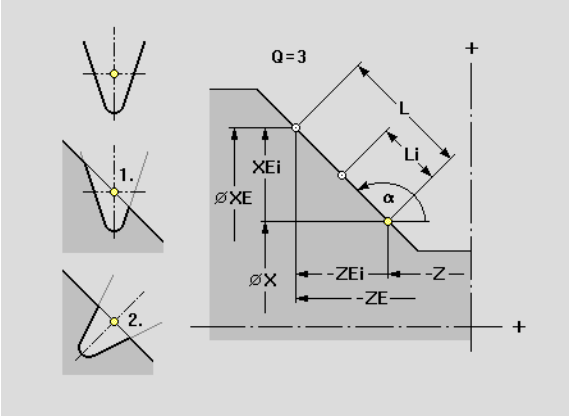
Reference point: Center of base element



Parameters

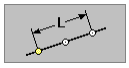
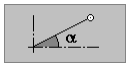
XF	Reference point shift
ZF	Reference point shift
R	■ R>0: Rounding radius ■ R=0: No rounding
A	Angular length
LS	Length of wedge sides (projecting element parts are cut at the “points of overlay”)
B	Width of base element
W	Angle of rotation: The overlay contour is rotated by the “angle of rotation.”

Linear superimposition (“linear overlay”)



Parameters

X	Starting point, position of the first overlay element
Z	Starting point, position of the first overlay element
	Position (see illustration) ■ 1: Original position: Inserts the overlay contour in the supporting contour “as is.” ■ 2: Normal position: Rotates the overlay contour about the slope angle of the supporting contour element and then inserts it in the supporting contour.
Q	Number of overlay elements
XE	End point, position of the last overlay element
ZE	End point, position of the last overlay element
XEi	Incremental end point
ZEi	Incremental end point
L	Distance between the first and last overlay element
Li	Distance between the overlay elements
a	Angle (default: Angle of the supporting contour element)

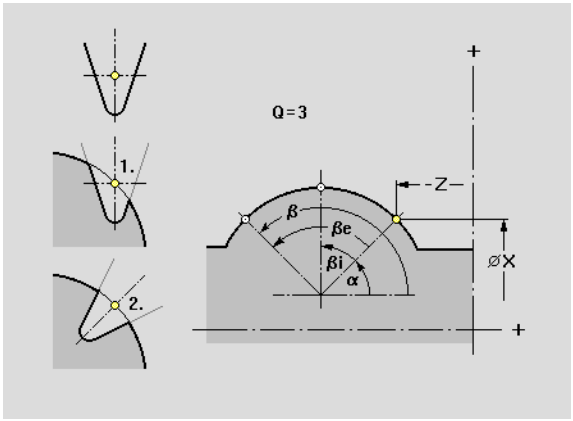
“Linear overlay” soft keys	
	Enter the length (instead of end point)
	Enter the angle

Circular superimposition (“circular overlay”)

The **direction of rotation** according to which the overlay contours are arranged corresponds to the direction of rotation of the supporting contour element.



The reference point of the contour to be superimposed is positioned on the “point of overlay.”



Parameters

X Starting point, position of the first overlay element

Z Starting point, position of the first overlay element

a Starting point as angle (reference: one line running parallel to the Z axis and through the center of the selected arc)

Position (see illustration)

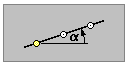
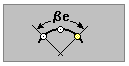
- 1: Original position: Inserts the overlay contour in the supporting contour “as is.”
- 2: Normal position: Rotates the overlay contour about the slope angle of the supporting contour element and then inserts it in the supporting contour.

Q Number of overlay elements

b End point, position of the last element to be superimposed (reference: a line segment running parallel to the Z axis and intersecting the center of the selected arc)

be Angle between the first and last overlay element

bi Angle between the individual overlay elements

“Circular overlay” soft keys	
	Angle of the first overlay position
	Angle of the last overlay position

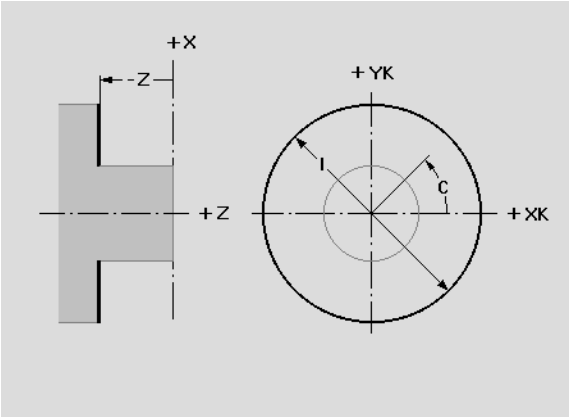
6.8 C-Axis Contours

Position of a front or rear face contour

TURN PLUS takes the selected reference plane and suggests it as reference dimension. Change the parameter, if required.

Parameters

Z Reference dimension

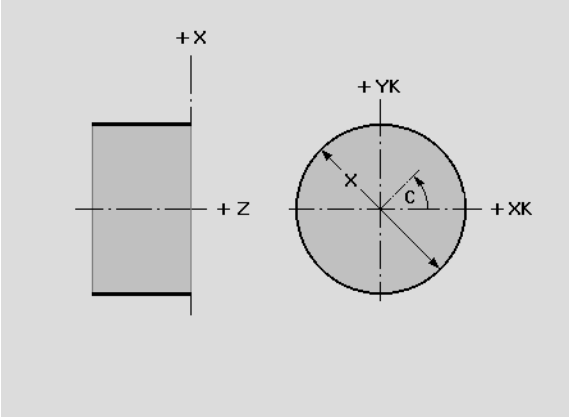


Position of a lateral surface contour

TURN PLUS takes the selected reference plane and suggests it as reference diameter. Change the parameter, if required.

Parameters

X Reference diameter



Milling depth

If you use individual elements for describing a milling contour, TURN PLUS opens the “Pocket/contour” dialog box after you have finished your entries, which requests “Depth P.”

Parameters

P Depth (P > 0 defines a pocket)

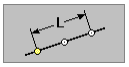
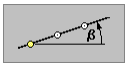
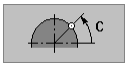
Entering the C-axis contour dimensions

Press the soft key for the type of dimensioning you want to use for the contour element, figure or pattern (see "Notes on defining contours" on page 404).

When defining **lateral surface contours**, enter either the angle or the "linear dimension." The linear dimension is given with respect to the unrolled surface at the "reference diameter."

Polar dimensions for lateral surface contours (parameter "P"):

- "P" is given with respect to the **unrolled lateral surface**.
- Select the desired solution when there are two possibilities.

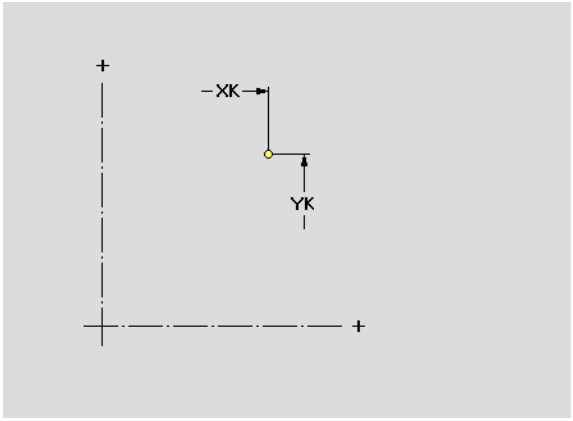
"Type of dimensioning" soft keys	
	Linear pattern: Enter the length
	Linear pattern: Enter the angle
	Lateral surface: Angle instead of linear dimension

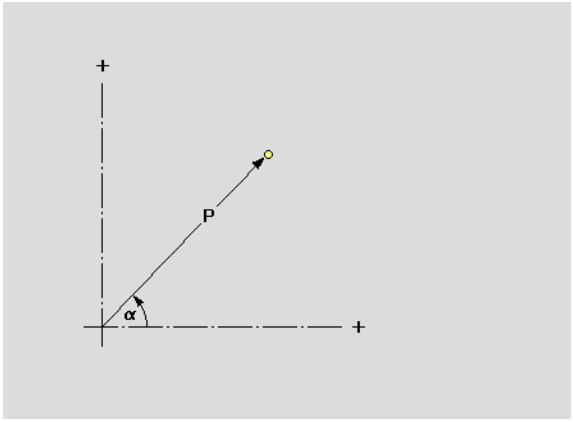
Front or rear face: Starting point

This function defines the starting point of a "free contour" on the front/rear face.

Parameters

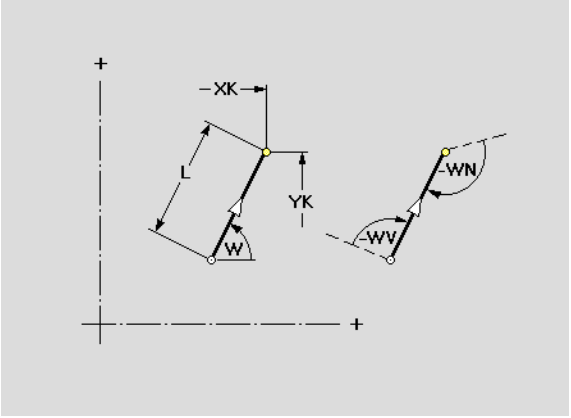
XK	Starting point of the contour in Cartesian coordinates
YK	Starting point of the contour in Cartesian coordinates
a	Starting point of the contour in polar coordinates (reference angle: positive XK axis)
P	Starting point of the contour in polar coordinates

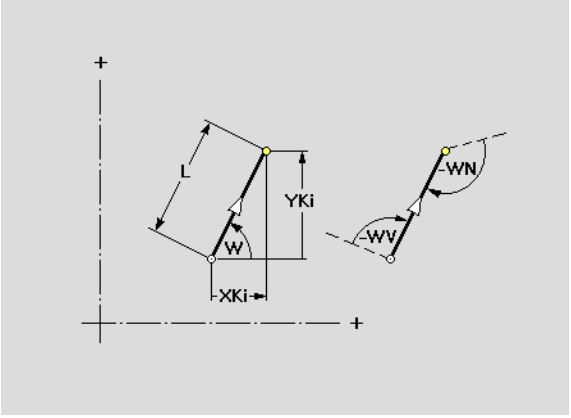


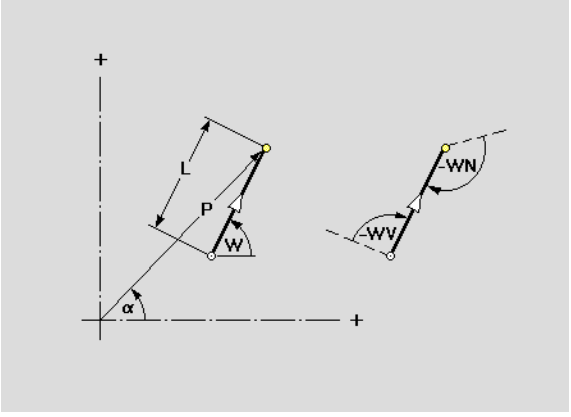


Front or rear face: Linear element

This function defines a line segment on the front/rear face.

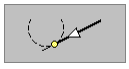






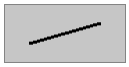
Parameters

XK	End point in Cartesian coordinates
YK	End point in Cartesian coordinates
XKi	Distance from starting point to end point
YKi	Distance from starting point to end point
a	End point in polar coordinates (reference angle: positive XK axis)
P	End point in polar coordinates
W	Angle of the line (for reference see illustration)
WV	Counterclockwise angle to the preceding element. If the preceding element is an arc: Angle to the tangent.
WN	Counterclockwise angle to the following element. If the following element is an arc: Angle to the tangent.
L	Length of element



Tangential/nontangential: Specify the transition to the next contour element

Defining a linear element:

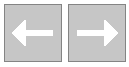


Call the lines menu.

Select the line direction.



Vertical line



Horizontal line



Line at angle



Line at angle

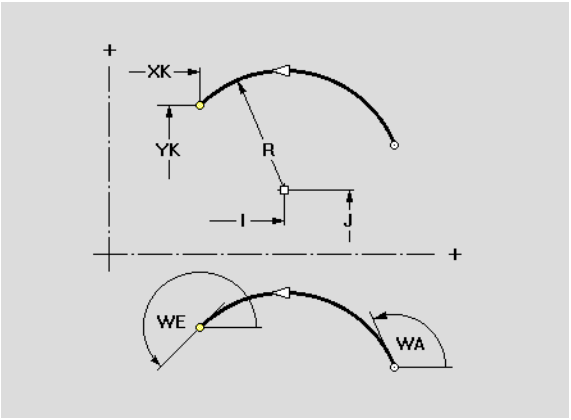


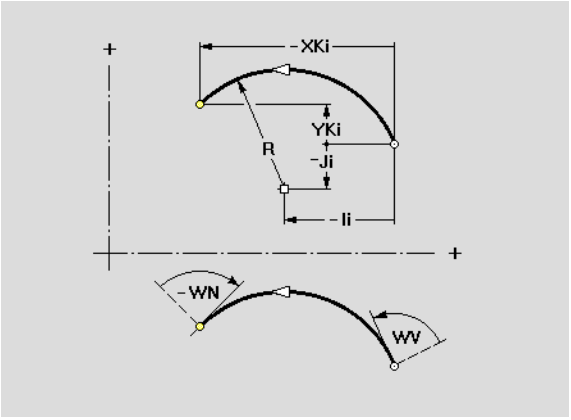
Line in any direction

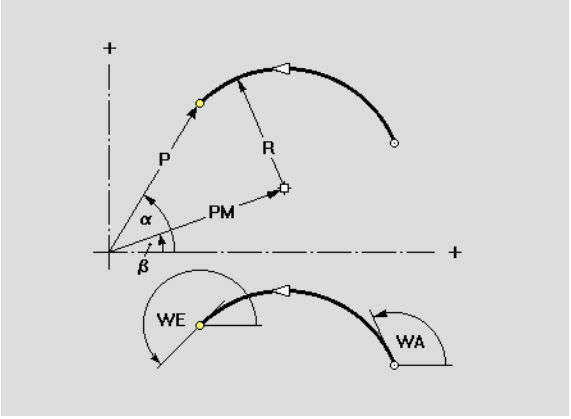
Enter the line dimensions and define the transition to the next element.

Front or rear face: Circular element

This function defines a circular arc on the front/rear face.







Parameters

End point of arc

XK	End point in Cartesian coordinates
YK	End point in Cartesian coordinates
XKi	Distance from starting point to end point
YKi	Distance from starting point to end point
a	End point in polar coordinates (reference angle: positive XK axis)
P	End point in polar coordinates
ai	End point polar, incremental (reference angle: Between an imaginary line intersecting the starting point and parallel to the XK axis, and another line from the starting point to the end point)
Pi	End point polar, incremental (Pi: linear distance from starting point to end point)

Center of arc

I	Center in Cartesian coordinates
J	Center in Cartesian coordinates
Ii	Distance from starting point to center point in XK direction
Ji	Distance from starting point to center point in YK direction
b	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
bi	Center point polar, incremental (reference angle: Angle between an imaginary line intersecting the starting point and parallel to the XK axis, and another line from the starting point to the center point)
PMi	Center point polar, incremental (linear distance from starting point to center point)

Parameters

Other parameters

R Arc radius



Tangential/nontangential: Specify the transition to the next contour element

WA Angle between positive XK axis and tangent in starting point of arc

WE Angle between positive XK axis and tangent in end point of arc

WV Counterclockwise angle between preceding element and tangent in starting point of arc. If the preceding element is an arc: Angle to the tangent.

WN Counterclockwise angle between tangent in arc end point and following element. If the following element is an arc: Angle to the tangent.

End point must not be the starting point (no full circle).

Defining a circular element:

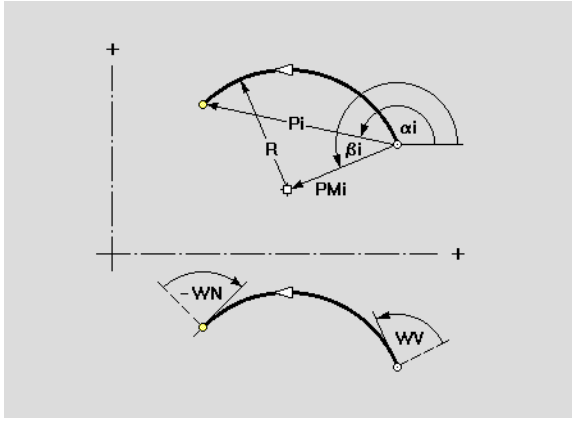


Call the arcs menu.



Select the direction of rotation.

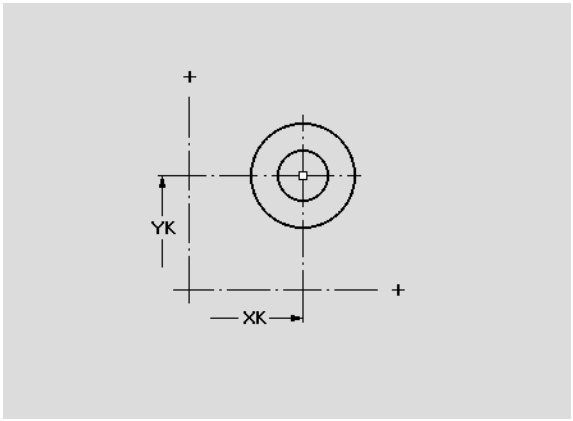
Enter the arc dimensions and define the transition to the next element.

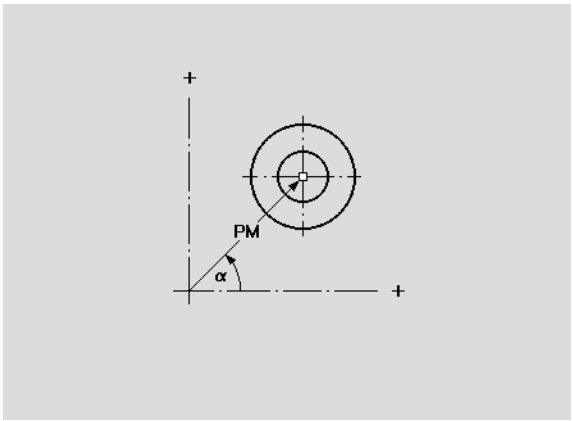


Front or rear face: Single hole

This function defines a single hole on the front/rear face. The hole may contain the following elements:

- Centering
- Core hole
- Countersinking
- Threads

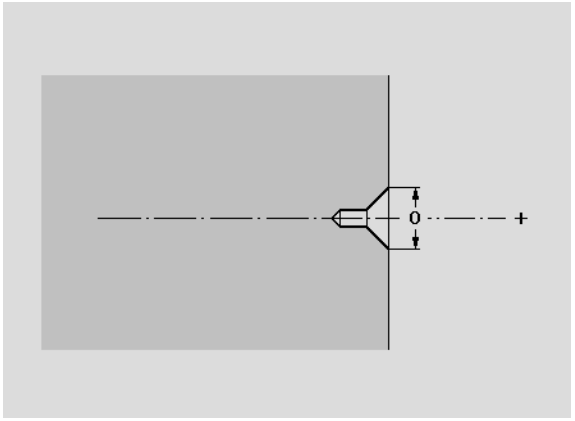




Hole reference point parameters

XK	Hole center in Cartesian coordinates
YK	Hole center in Cartesian coordinates
a	Hole center in polar coordinates (reference angle: positive XK axis)
PM	Hole center in polar coordinates

Centering on front/rear face contour



Centering parameters

Q	Centering diameter

Core hole on front/rear face contour

Core hole parameters

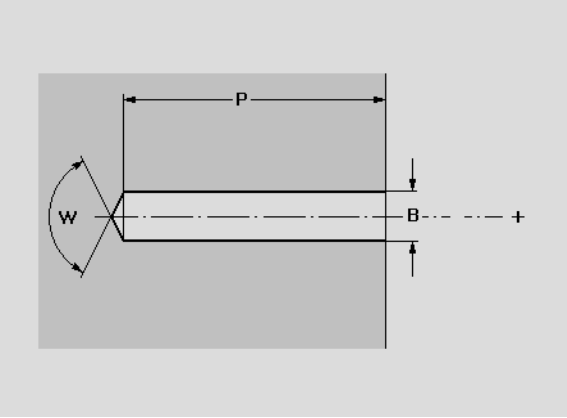
B Diameter of hole

P Depth of hole (excluding point)

W Point angle

- W=0°: the AWG generates a "feed rate reduction (V=1)" for the drilling cycle
- W>0°: point angle

Fit: H6 to H13 or "without fit" (see "Drilling" on page 556)



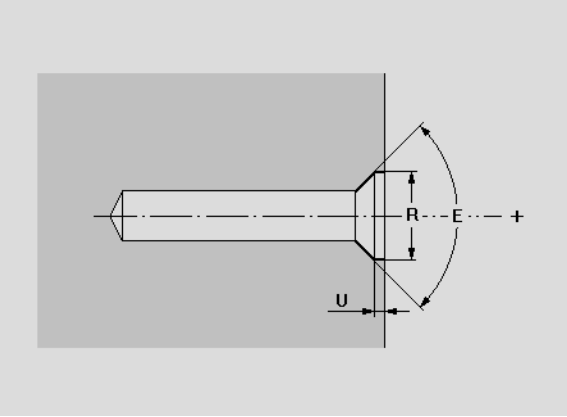
Countersinking on front/rear face contour

Countersinking parameters

R Sinking diameter

U Sinking depth

E Sinking angle



Tapping on front/rear face contour

Thread parameters

I Nominal diameter

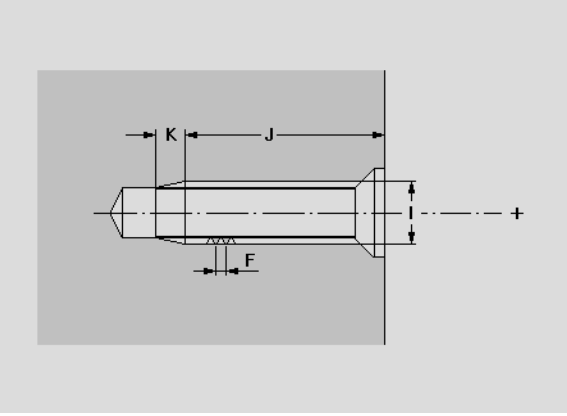
J Thread depth

K Thread runout length

F Thread pitch

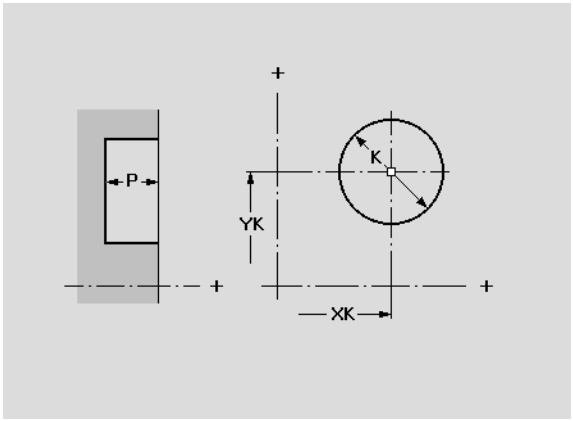
Type of thread:

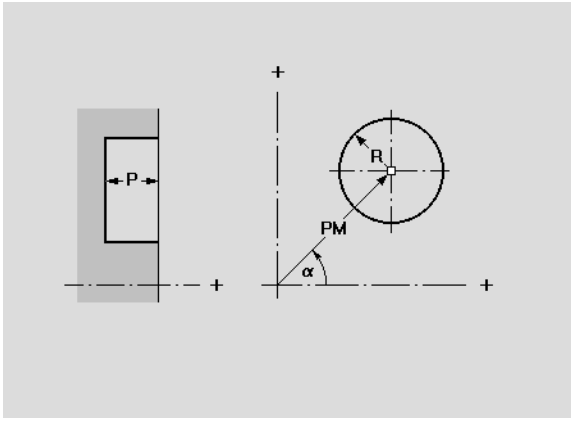
- Right-hand thread
- Left-hand thread



Front or rear face: Circle (full circle)

This function defines a full circle on the front/rear face.



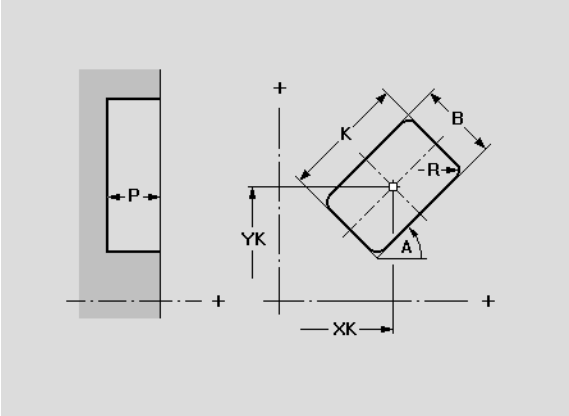


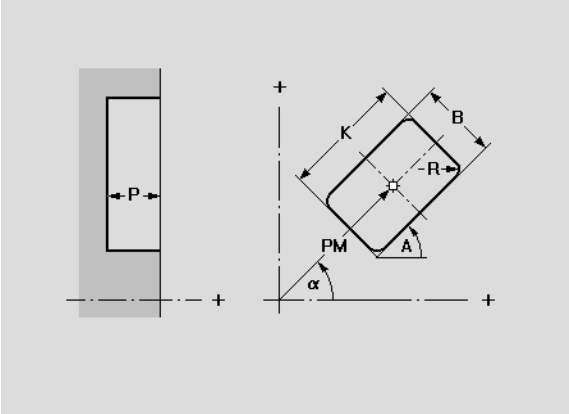
Parameters

XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
a	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
R	Circle radius
K	Circle diameter
P	Depth of the figure

Front or rear face: Rectangle

This function defines a rectangle on the front/rear face.



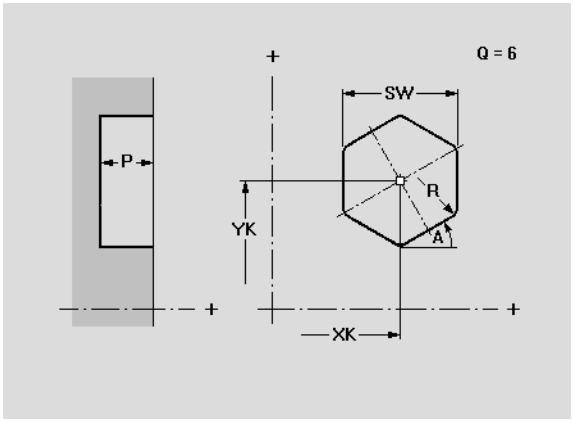


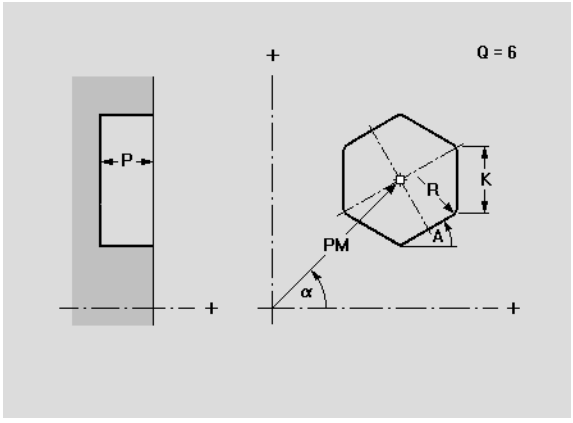
Parameters

XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
a	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
A	Angle to longitudinal axis of rectangle (reference: XK axis)
K	Length of rectangle
B	Width of rectangle
R	Chamfer/rounding ■ Width of chamfer ■ Radius of rounding
P	Depth of the figure

Front or rear face: Polygon

This function defines a polygon on the front/rear face.



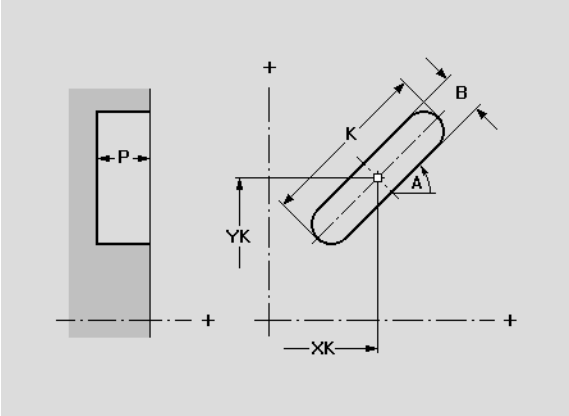


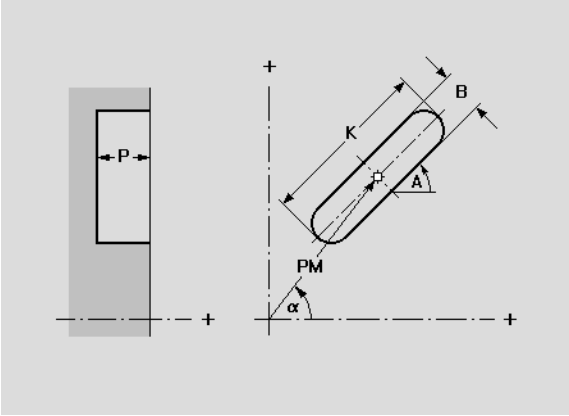
Parameters

XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
a	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
A	Angle to a polygon side (reference: XK axis)
Q	Number of corners (Q>=3)
K	Edge length
SW	Width across flats (inscribed circle diameter)
R	Chamfer/rounding ■ Width of chamfer ■ Radius of rounding
P	Depth of the figure

Front or rear face: Linear slot

This function defines a linear slot on the front/rear face.



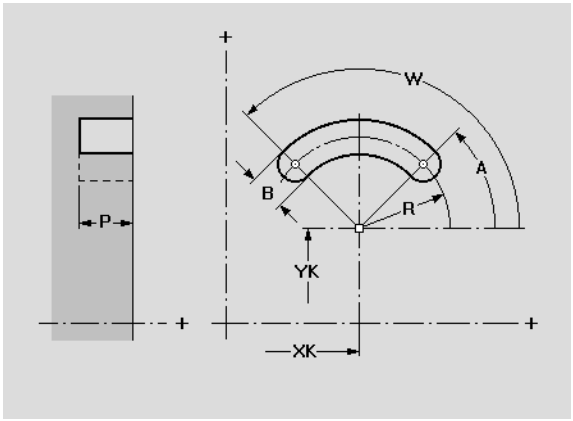


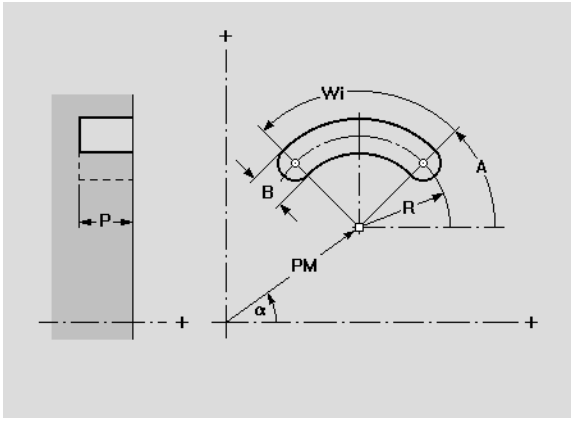
Parameters

XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
a	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
A	Angle of longitudinal axis of slot (reference: XK axis)
K	Slot length
B	Slot width
P	Depth of the figure

Front or rear face: Circular slot

This function defines a circular slot on the front/rear face.





Parameters

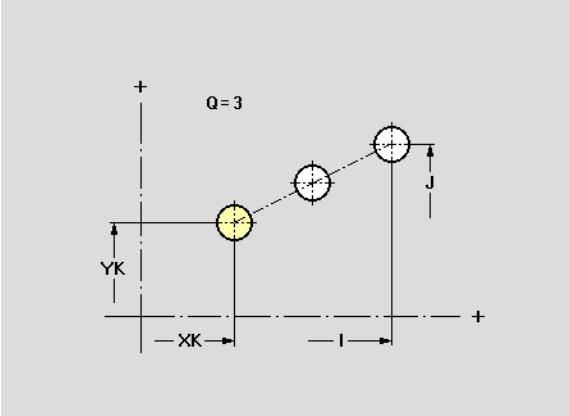
XK	Center in Cartesian coordinates
YK	Center in Cartesian coordinates
a	Center point in polar coordinates (reference angle: positive XK axis)
PM	Center point in polar coordinates
A	Starting angle (starting point) of the slot (reference: XK axis)
W	End angle (end point) of the slot (reference: XK axis)
R	Curvature radius (reference: center point path of the slot)
B	Slot width
P	Depth of the figure

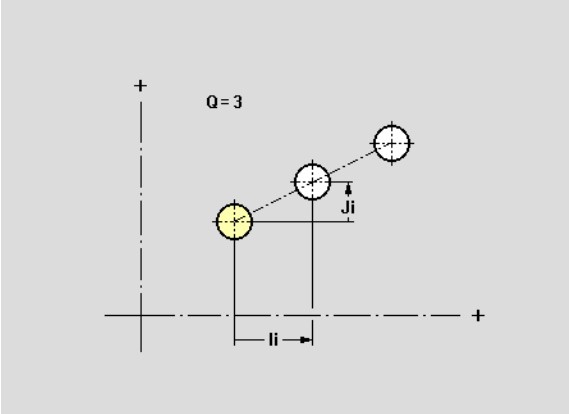
Front or rear face: Linear hole or figure pattern

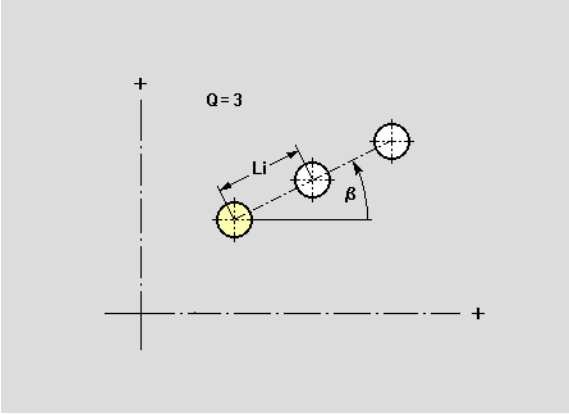
This function defines a linear hole or figure pattern on the front/rear face.

Parameters

XK	Starting point of pattern in Cartesian coordinates
YK	Starting point of pattern in Cartesian coordinates
a	Starting point of pattern in polar coordinates (reference angle: positive XK axis)
P	Starting point of pattern in polar coordinates
Q	Number of figures (default: 1)
I	End point of pattern in Cartesian coordinates
J	End point of pattern in Cartesian coordinates
Ii	Distance in XK direction between two figures
Ji	Distance in YK direction between two figures
b	Angle to the longitudinal axis of the pattern (reference: XK axis)
L	Total length of pattern
Li	Distance between two figures (pattern distance)

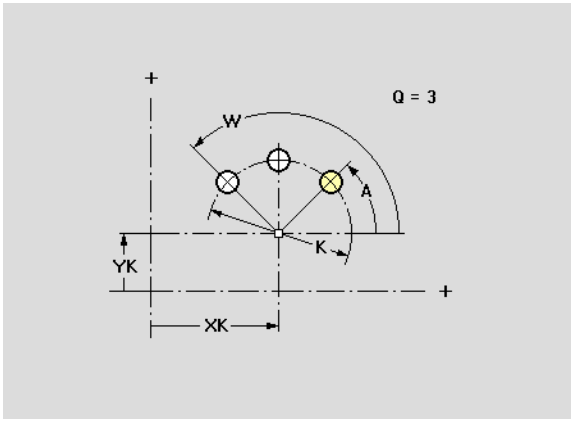


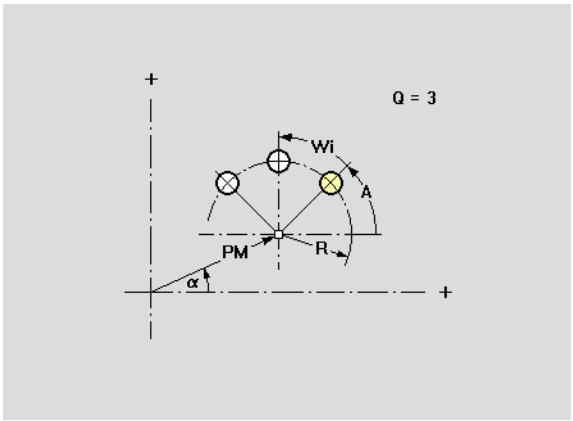




Front or rear face: Circular hole or figure pattern

This function defines a circular hole or figure pattern on the front/rear face.





Parameters

XK Center of pattern in Cartesian coordinates

YK Center of pattern in Cartesian coordinates

a Center of pattern in polar coordinates (reference angle: positive XK axis)

PM Center of pattern in polar coordinates

Q Number of figures

Orientation:

- Clockwise
- Counterclockwise

R Radius of the pattern

K Diameter of the pattern

A Starting angle, position of the first figure (reference: XK axis)

If A and W are not programmed: Figures are arranged on a full circle, starting at 0°

W End angle, position of the last figure (reference: XK axis)

If W is not programmed: Figures are arranged on a full circle, starting at A

Wi Angle between two figures (algebraic sign has no effect)

Position of the figures

- Normal position: The original figure is rotated about the center of the pattern (rotation)
- Original position: The position of the original figure is maintained (translation)

Hole description/figure description



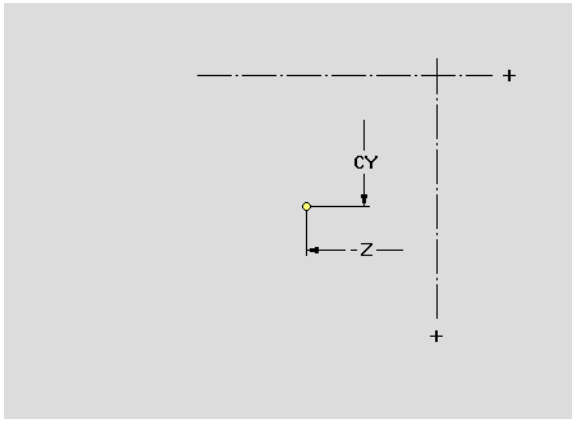
When defining patterns with circular slots, the "center of curvature" is added to the pattern position (see "Circular pattern with circular slots" on page 169).

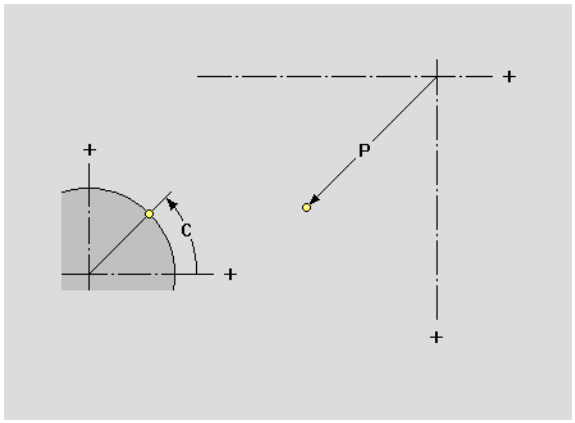
Lateral surface: Starting point

This function defines the starting point of a “free contour” on the lateral surface.

Parameters

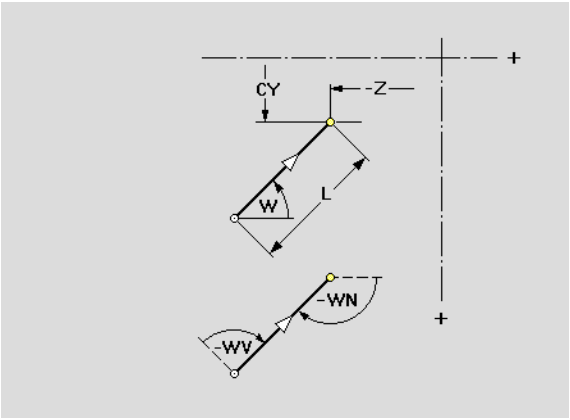
Z	Starting point of contour
P	Starting point of contour – polar
CY	Starting point of contour – angle as “linear dimension”
C	Starting point of contour – angle

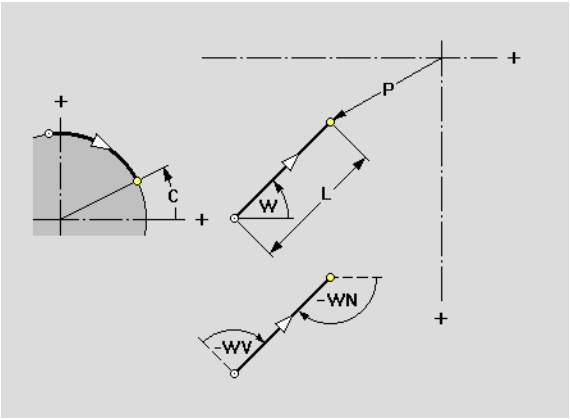




Lateral surface: Linear element

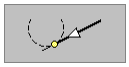
This function defines a line segment on a lateral surface.





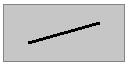
Parameters

Z	End point of the line
P	End point of the line – polar
CY	End point of the line – angle as "linear dimension"
C	End point – angle
W	Angle of the line (for reference see illustration)
WV	Counterclockwise angle to the preceding element. If the preceding element is an arc: Angle to the tangent.
WN	Counterclockwise angle to the following element. If the following element is an arc: Angle to the tangent.
L	Length of element



Tangential/nontangential: Specify the transition to the next contour element

Defining a linear element:

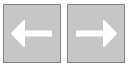


Call the lines menu.

Select the line direction.



Vertical line



Horizontal line



Line at angle



Line at angle

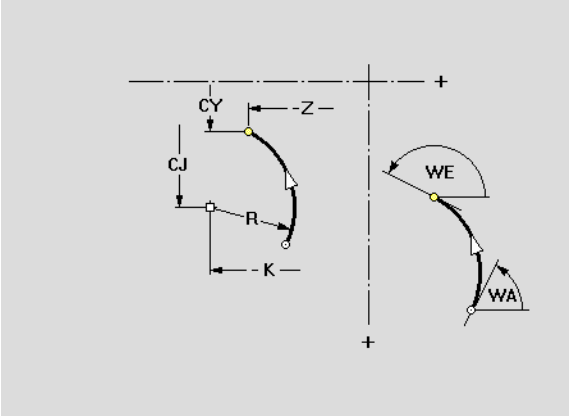


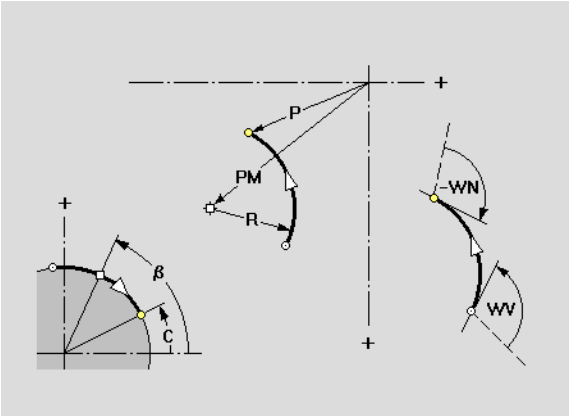
Line in any direction

Enter the line dimensions and define the transition to the next element.

Lateral surface: Circular element

This function defines a circular arc on a lateral surface.





Parameters

End point of arc

Z End point

P End point – polar

CY End point – angle as “linear dimension”

C End point – angle

Center of arc

K Center

CJ Center point (angle as “linear dimension”)

b Center point in polar coordinates (reference angle: positive XK axis)

PM Center point – polar

Other parameters

R Arc radius



Tangential/nontangential: Specify the transition to the next contour element

WA Angle between positive Z axis and tangent in starting point of arc

WE Angle between positive Z-axis and tangent in end point of arc

WV Counterclockwise angle between preceding element and tangent in starting point of arc. If the preceding element is an arc: Angle to the tangent.

WN Counterclockwise angle between tangent in arc end point and following element. If the following element is an arc: Angle to the tangent.



Call the arcs menu.



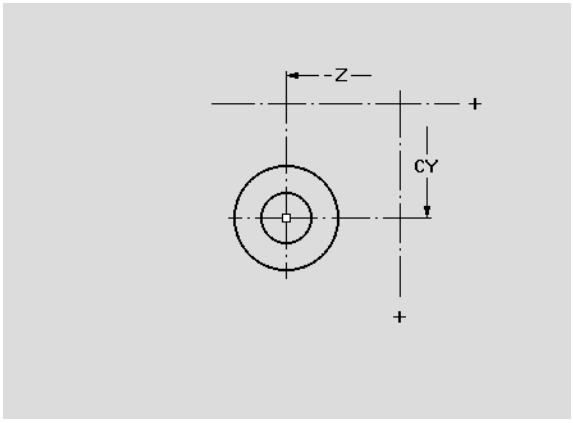
Select the direction of rotation.

Enter the arc dimensions and define the transition to the next element.

Lateral surface: Single hole

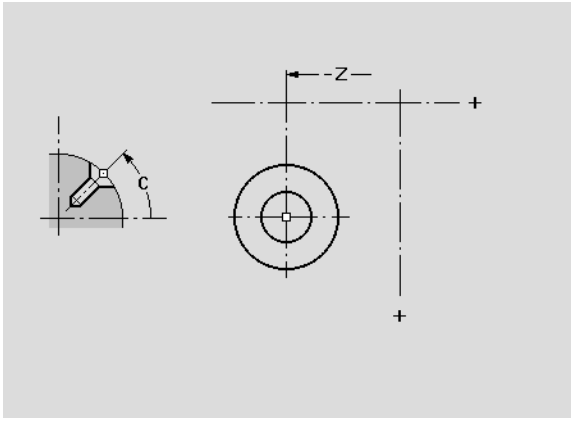
This function defines a single hole on the lateral surface. The hole may contain the following elements:

- Centering
- Core hole
- Countersinking
- Threads



Hole reference point parameters

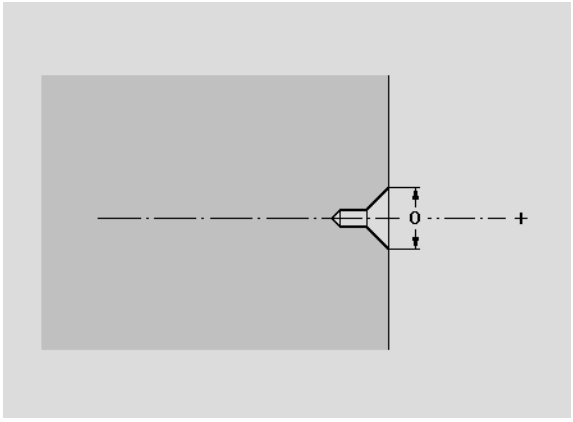
Z	Center of hole
CY	Center of hole – angle as “linear dimension”
C	Center of hole – angle



Centering on lateral surface contour

Centering parameters

Q	Centering diameter



Core hole on lateral surface contour

Core hole parameters

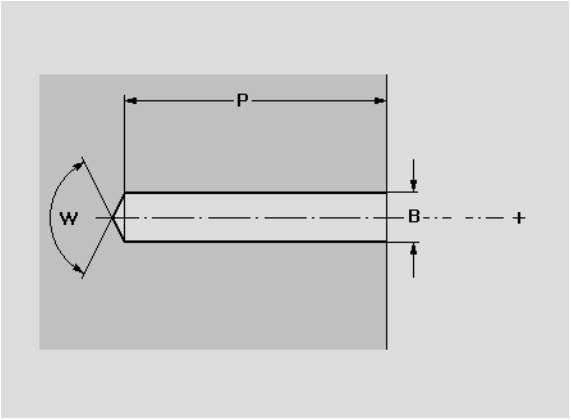
B Diameter of hole

P Hole depth (depth of hole and countersink – without drill point and center point)

W Point angle

- W=0°: the AWG generates a "feed rate reduction (V=1)" for the drilling cycle
- W>0°: point angle

Fit: H6 to H13 or "without fit" (see "Drilling" on page 556)



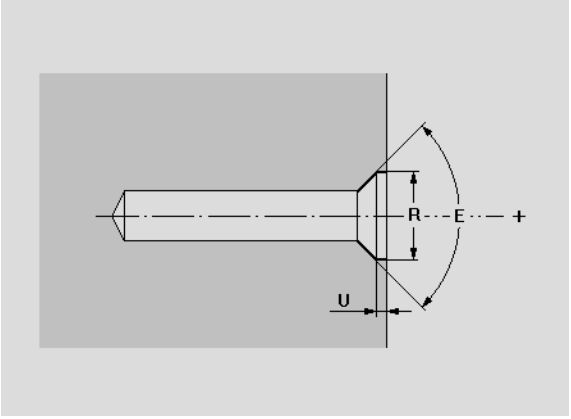
Countersinking on lateral surface contour

Countersinking parameters

R Sinking diameter

U Sinking depth

E Sinking angle



Tapping on lateral surface contour

Thread parameters

I Nominal diameter

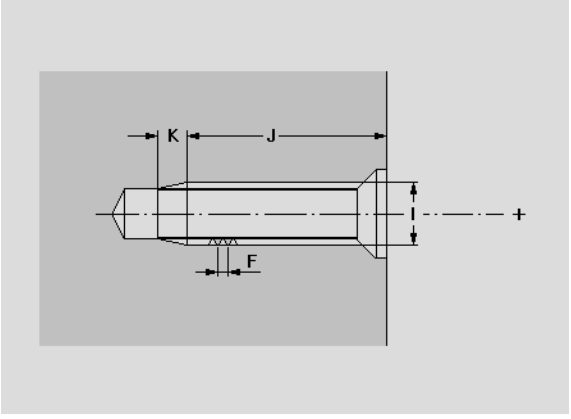
J Thread depth

K Thread runout length

F Thread pitch

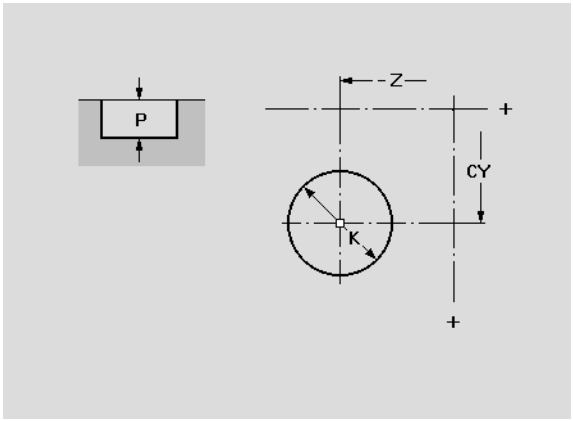
Type of thread:

- Right-hand thread
- Left-hand thread



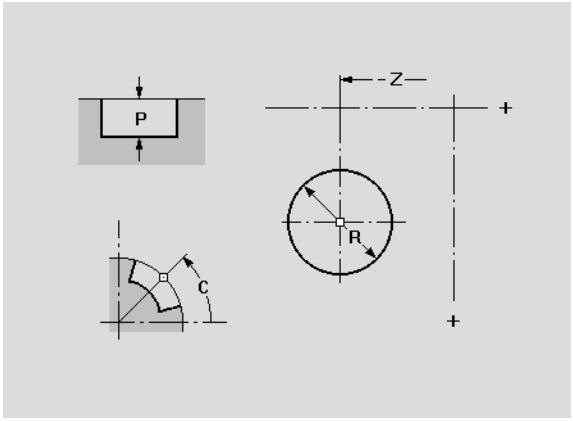
Lateral surface: Circle (full circle)

This function defines a full circle on the lateral surface.



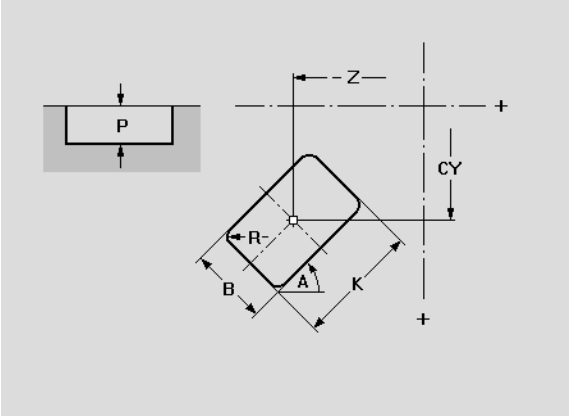
Parameters

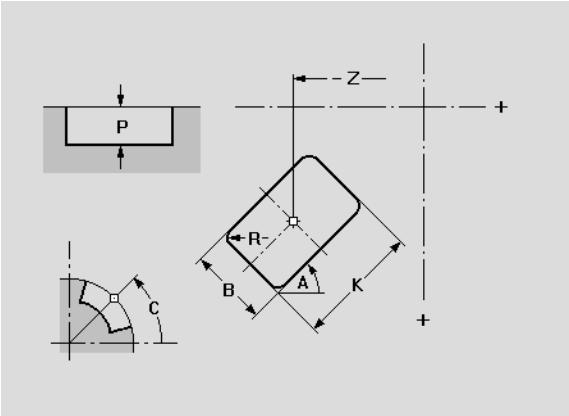
Z	Center of the figure
CY	Center of the figure – angle as “linear dimension”
C	Center of the figure – angle
R	Radius
K	Circle diameter
P	Depth of the figure



Lateral surface: Rectangle

This function defines a rectangle on the lateral surface.



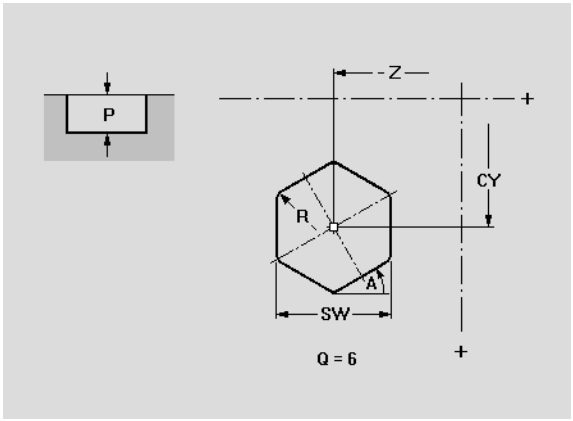


Parameters

Z	Center of the figure
CY	Center of the figure – angle as “linear dimension”
C	Center of the figure – angle
A	Angle to longitudinal axis of rectangle (reference Z axis)
K	Length of rectangle
B	Width of rectangle
R	Chamfer/rounding ■ Width of chamfer ■ Radius of rounding
P	Depth of the figure

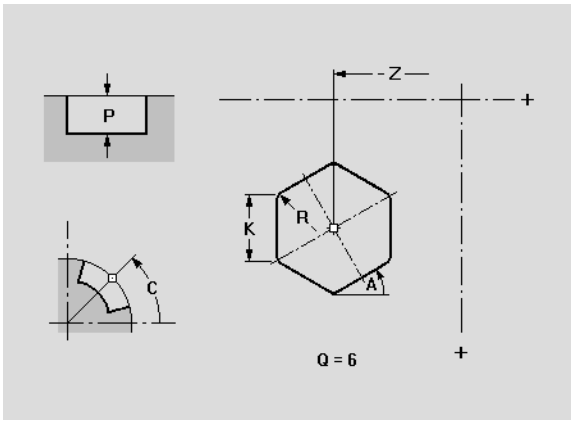
Lateral surface: Polygon

This function defines a polygon on the lateral surface.



Parameters

Z	Center of the figure
CY	Center of the figure – angle as “linear dimension”
C	Center of the figure – angle
A	Angle to a polygon side (reference: Z axis)
Q	Number of corners (Q>=3)
K	Edge length
SW	Width across flats (inscribed circle diameter)
R	Chamfer/rounding ■ Width of chamfer ■ Radius of rounding
P	Depth of the figure

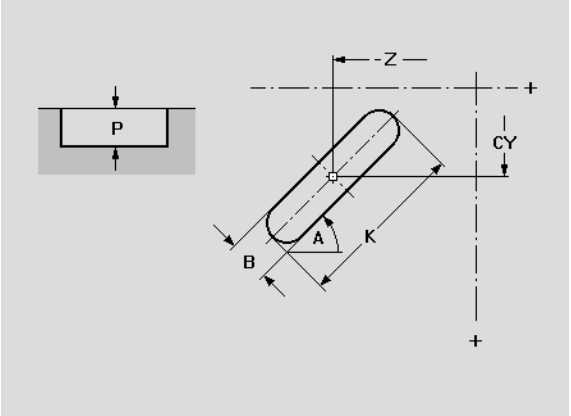


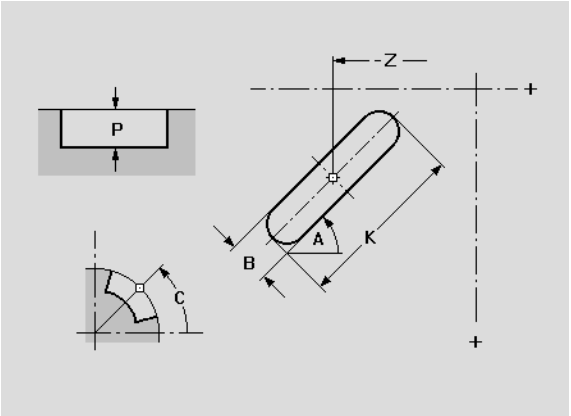
Lateral surface: Linear slot

This function defines a linear slot on the lateral surface.

Parameters

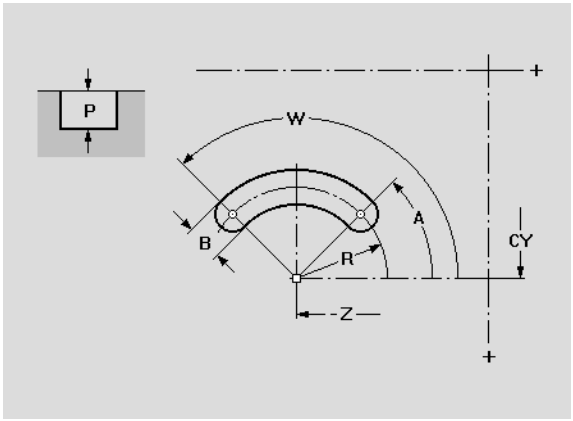
Z	Center of the figure
CY	Center of the figure – angle as “linear dimension”
C	Center of the figure – angle
A	Angle to longitudinal axis of slot (reference: Z axis)
K	Slot length
B	Slot width
P	Depth of the figure





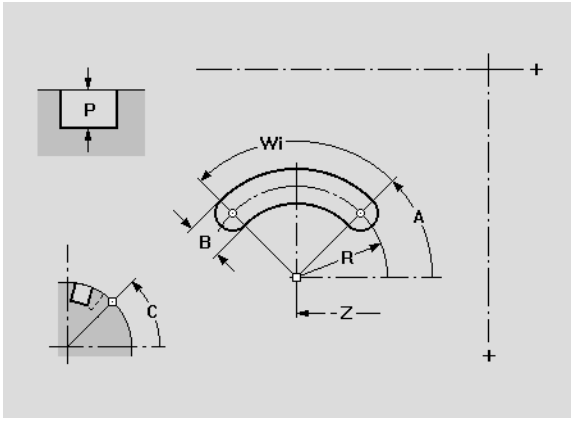
Lateral surface: Circular slot

This function defines a circular slot on the lateral surface.



Parameters

Z	Center of the figure
CY	Center of the figure – angle as "linear dimension"
C	Center of the figure – angle
A	Starting angle (starting point) of the slot (reference: Z axis)
W	End angle (end point) of the slot (reference: Z axis)
B	Slot width
P	Depth of the figure

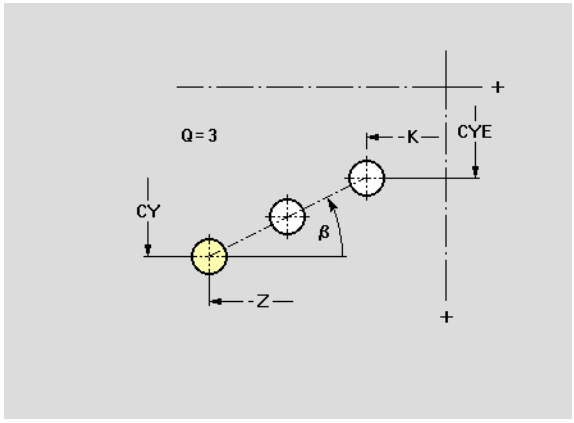


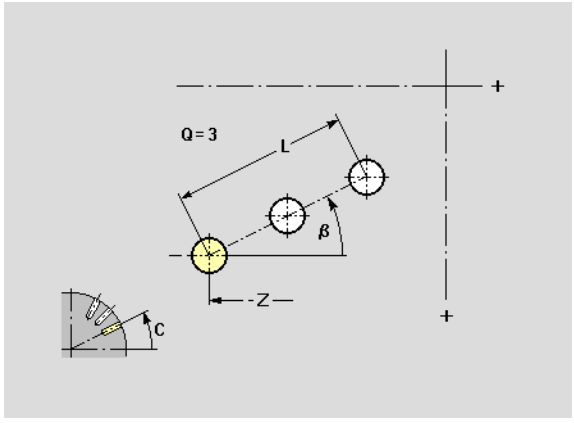
Lateral surface: Linear hole or figure pattern

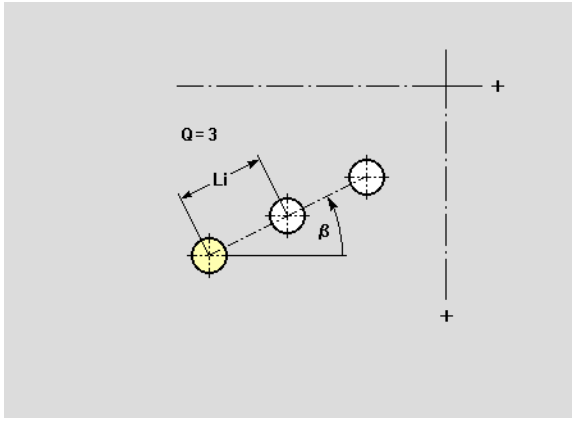
This function defines a linear hole or figure pattern on the lateral surface.

Parameters

Z	Starting point of pattern
CY	Starting point of pattern – angle as “linear dimension”
C	Starting point of pattern – angle
Q	Number of figures (default: 1)
K	End point of pattern
Ki	Distance in Z direction between two figures
CYE	End point of pattern – angle as “linear dimension”
CYi	Distance between figures – as “linear dimension”
L	Total length of pattern
Li	Distance between two figures (pattern distance)
b	Angle to longitudinal axis of pattern (reference: Z axis)
	Hole description/figure description



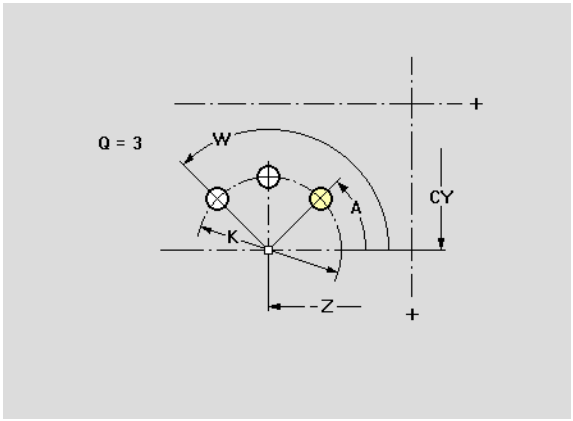


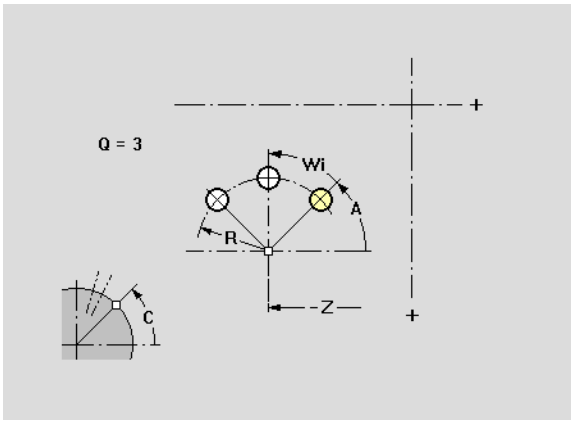


If you do not program the end point, the holes/figures will be distributed evenly along the circumference.

Lateral surface: Circular hole or figure pattern

This function defines a linear hole or figure pattern on the lateral surface.





Parameters

Z Center of pattern

CY Center of pattern – angle as "linear dimension"

C Center of pattern – angle

Q Number of figures (default: 1)

Orientation

- Clockwise
- Counterclockwise

R Radius of the pattern

K Diameter of the pattern

A Starting angle, position of the first figure (reference: Z axis)

If A and W are not programmed: Figures are arranged on a full circle, starting at 0°

W End angle, position of the last figure (reference: Z axis)

If W is not programmed: Figures are arranged on a full circle, starting at A

Wi Angle between two figures (algebraic sign has no effect)

Position of the figures

- Normal position: The original figure is rotated about the center of the pattern (rotation)
- Original position: The position of the original figure is maintained (translation)

Hole description/figure description



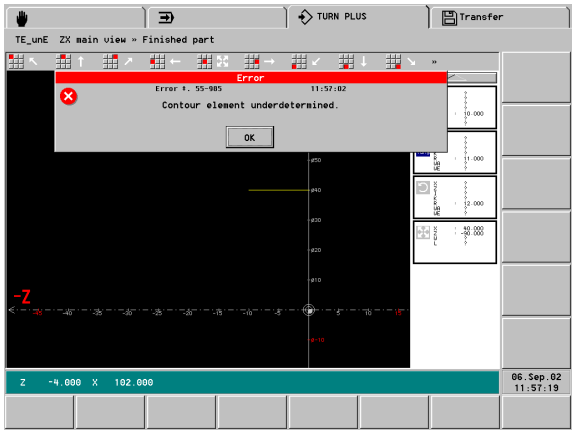
When defining patterns with circular slots, the "center of curvature" is added to the pattern position (see "Circular pattern with circular slots" on page 169).

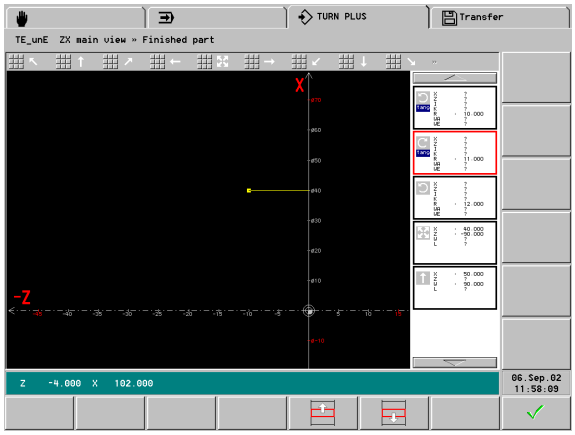
6.9 Help Functions

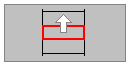
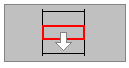
Unresolved contour elements

Contour elements that cannot be calculated are referred to as “unresolved elements.” TURN PLUS displays these elements on the right side of the screen. Each unresolved element is depicted by a symbol. TURN PLUS additionally lists all the parameters that are already known.

If a contour element is not sufficiently defined, TURN PLUS reports the error. After you have confirmed the error message, use the soft keys to position the cursor to the desired unresolved element and correct the data.





Soft keys	
	Mark previous unresolved element
	Mark next unresolved element
	Select marked unresolved element

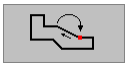
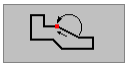
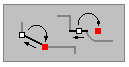
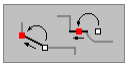
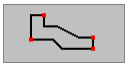
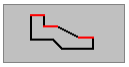
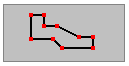
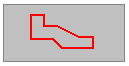
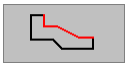
Selections

Contour points or contour elements are first selected and then superimposed by form elements.



Colors of selection points

- Red: Point marked by cursor, but not selected
- Green: Selected point
- Blue: Point marked by cursor and selected

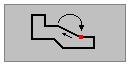
Selection soft keys	
	Next contour point (alternative: “Arrow left”)
	Previous contour point (alternative: “Arrow right”)
	Next contour element (alternative: “Arrow left”)
	Previous contour element (alternative: “Arrow right”)
	Previous position for hole (alternative: “Arrow left”)
	Next position for hole (alternative: “Arrow right”)
	Enable multiple selection of contour points
	Enable multiple selection of contour elements
	Select all contour points
	Select all contour elements
	Activate area selection.
	■ Select the contour point/contour element. ■ Conclude selection
	Cancel the contour point/contour element selection

Selecting a single contour point/contour element
Single selection by touch pad

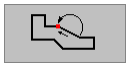
Place the cursor on the contour point or contour element.

Press the left mouse button – the contour point/element is selected.

Single selection by soft key

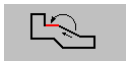


Go to the contour point.





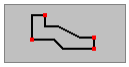
Go to the contour element.



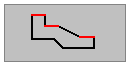


Select the contour point/contour element.

Selecting more than one contour point/element
Multiple selection by touch pad



Activate multiple selection of contour points.

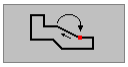


Activate multiple selection of contour elements.

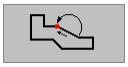
For each contour point/element you want to select:

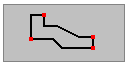
Place the cursor on the contour point/element and press the left mouse button.

Multiple selection by soft key

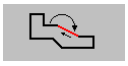


Go to the first contour point.



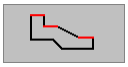


Mark the contour point and activate multiple selection.



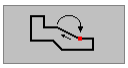
Go to the first contour element.



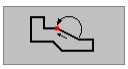


Mark the contour element and activate multiple selection.

For each contour point/element you want to select:



Go to the contour point.





Go to the contour element.





Mark the contour point/contour element.



Conclude the selection.

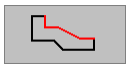


Alternatively, you can select all contour points/contour elements and then cancel the points/elements selections that are not needed.

Selecting a contour area

Area selection by touch pad

Place the cursor on the first element.



Activate area selection.

Place the cursor on the last element.

Press the left mouse button: Area is selected in the direction of contour definition

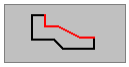
Press the right mouse button: Area is selected opposite to the direction of contour definition

Area selection by soft key



Go to the beginning of the area.





Mark the beginning of the area and activate area selection.



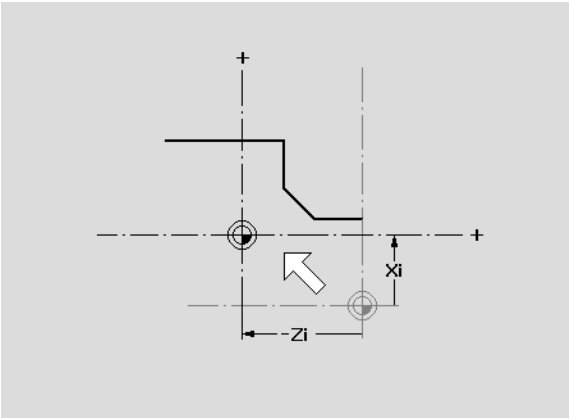
Go to the end of the area.





Conclude area selection.

Zero point shift



Example: If the workpiece dimensions refer to different sides, you start by defining the contour elements whose dimensions are referenced to the right-hand side. Then shift the zero point and define the contour elements whose dimensions are referenced to the left-hand side.

Activate zero point shift:

- Select “Zero point > Shift” in the finished part menu. TURN PLUS opens the “Shift zero point” dialog box.
- Enter the zero point shift. TURN PLUS shifts the contour you have defined so far.

Deactivate zero point shift:

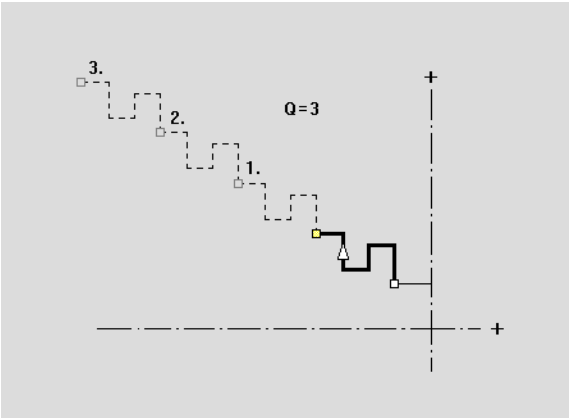
- Select “Zero point > Reset“ in the finished part menu. TURN PLUS resets the zero point of the coordinate system to the originally programmed position.

Parameters

Xi Target point – value by which the zero point is shifted.

Zi Target point – value by which the zero point is shifted.

Copying a contour section in linear series



With this function, you can define a contour section and append it to the existing contour any number of times.

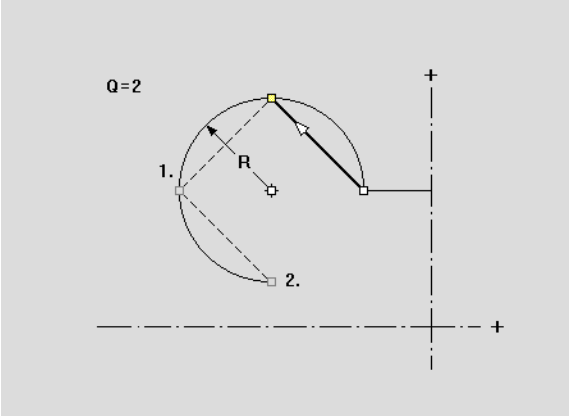
- Select “Copy > Row > Linear” in the finished part menu. TURN PLUS marks the last element.
- Select the contour section (you can only select the last contour elements entered).
- TURN PLUS opens the “Copy in linear series” dialog box. Enter the number of copies you want to append.
- TURN PLUS extends the contour.

Parameters

Q Number of copies (the contour section is copied Q times)

Copying a contour section in circular series

With this function, you can define a contour section and append it to the existing contour any number of times.



- Select “Copy > Row > Circular” in the finished part menu. TURN PLUS marks the last element.
- Select the contour section (you can only select the last contour elements entered).
- TURN PLUS opens the “Copy in circular series” dialog box. Enter the number of copies and the radius.
- TURN PLUS displays the first center of rotation as a red square. Select the appropriate center of rotation.
- TURN PLUS extends the contour.

Parameters

Q Number of copies (the contour section is copied Q times)

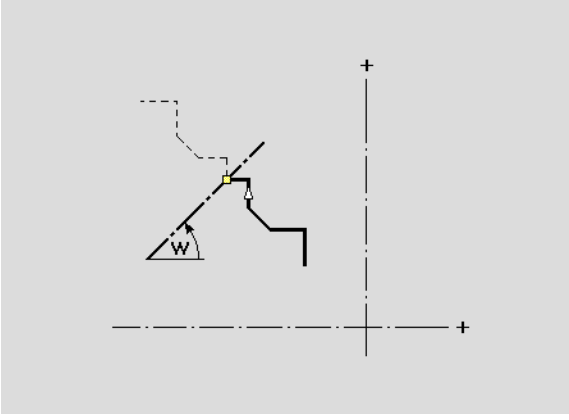
R Radius

Execution of “circular copying”

- **Centers of rotation:** TURN PLUS uses the entered radius for creating a circle both around the starting point and the end point of the contour section. The points of intersection of the circles are the two possible centers of rotation.
- The **angle of rotation** is calculated from the distance between starting point and end point of the contour section.
- **Extending the contour:** TURN PLUS copies the selected contour section, rotates it and adds it to the contour.

Copying a contour section by mirroring

With this function, you can define a contour section, mirror it and add it to the existing contour.



- Select “Copy > Mirror” in the finished part menu. TURN PLUS marks the last element.
- Select the contour section (you can only select the last contour elements entered).
- TURN PLUS opens the “Copy by mirroring” dialog box.
- Define the mirror axis. TURN PLUS extends the contour.

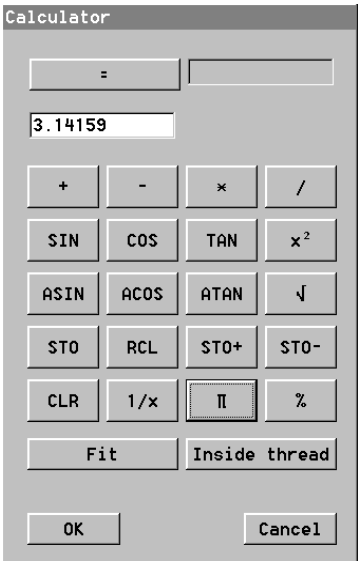
Parameters

W Angle of the mirror axis. The mirror axis intersects the current end point of the contour.

Reference of the angle: positive Z axis

Calculator

You can use an online pocket calculator for example for standard calculations, calculation of fit tolerances, and calculation of the core hole diameter for inside threads.



Calculate:

- Position the cursor on the input field of the dialog box



- Call the calculator. The value of the input box is loaded.
- Perform the calculation.
- "OK" closes the calculator and applies the values.
- "Cancel" closes the calculator without applying the values.

Operation:

- Select and activate the function/box by cursor key or mouse.
- The arithmetical functions (SIN, Square, etc.) refer to the value displayed.

Displays:

- Value displayed (below "=")
- Saved value (to the right of "=")
- Arithmetic operation and subtotal (to the right of the value displayed)

Calculate **fit** (calculates the mean tolerance for fits)

- Enter the nominal diameter.
- Press "Fit."
- Enter the data of the fit ("Fit" dialog box).
- Press "OK." The calculator displays the mean tolerance value.

Calculate **core hole diameter** for inside threads (the diameter is derived from the thread data):

- Press "Inside thread."
- Enter the thread data ("Inside thread" dialog box).
- Press "OK." The calculator calculates the core hole diameter and accepts it as display value.

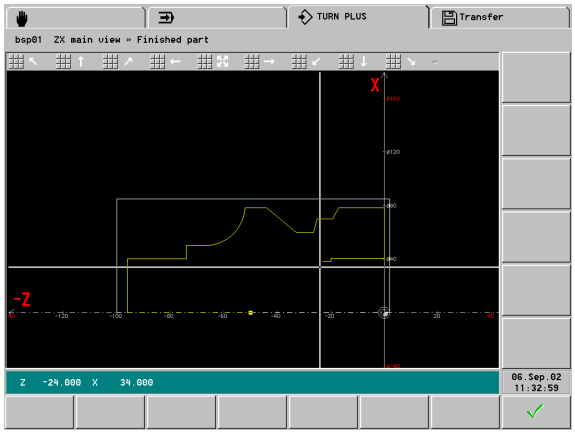
Calculator functions	
=	Perform calculation; display result
+,-,*,/	Basic operations
SIN, COS, TAN	Trigonometric functions
ASIN, ACOS, ATAN	Inverse trigonometry functions
X^2	Square
÷	Square root
STO	Store value displayed
STO+	Add value displayed to memory contents
STO–	Subtract value displayed from memory contents
RCL	Recall memory contents to the display
CLR	Clear display
1/X	Calculate the reciprocal of the value displayed
p	Ratio of circle circumference to diameter (3.14159)
n %	Percent calculation

Digitizing

With the digitizing feature, you determine the input values using the quadruple arrow (cross hairs) and transfer them to the input field. TURN PLUS displays the coordinates of the cross-hairs position.



- Activate digitizing mode with the dialog box open.
- Position the cross hairs by arrow keys or touch pad.
- Exit the digitizing mode:
  - ENTER key: Values are applied.
  - ESC key: Values are not applied.



- Change the zoom setting **before** calling the digitizing mode if the increments of the cross hairs movements are too small/large.
- The values are taken over as **absolute values** of the Cartesian coordinate system, independent from the setting of the input fields.

Checking contour elements (inspector)

The “inspector function” can be used to check contour elements, form elements, figures and patterns. It is not possible to edit the displayed data.

Select the desired window (reference plane).



Activate the magnify function.



Call the “inspector.”

Place the cursor on the contour element, form element, figure or pattern.



Confirm the position. TURN PLUS shows the **entered** parameters.

Press the ALT key: TURN PLUS shows **all** parameters of the element - for form elements the parameters of the individual elements.

Press the arrow left/right (with opened dialog box): TURN PLUS shows the parameters of the following/previous element.



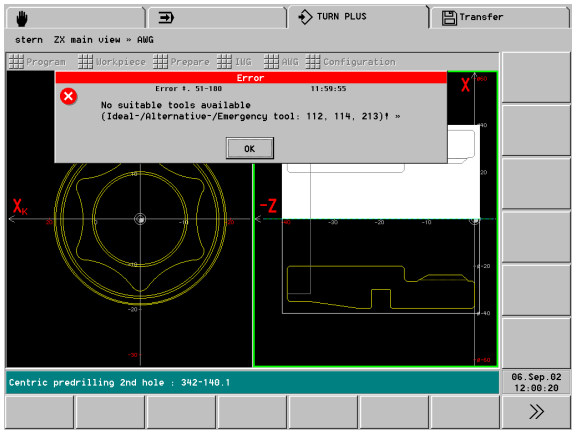
Press the ESC key: Close the dialog box.

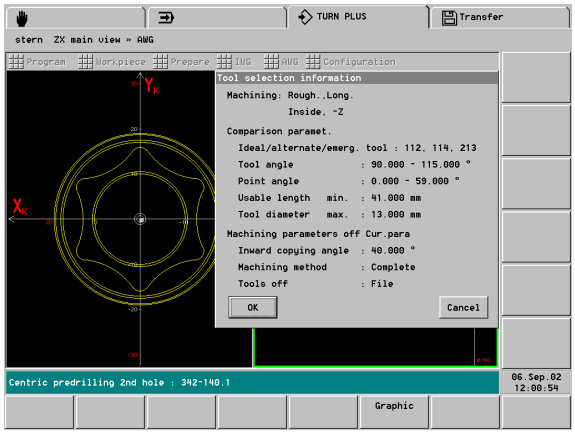
Error messages

If the actual error message shows the characters “>>,” TURN PLUS can display further information on the error, if desired.



▶ Display additional information on the error.





6.10 Importing of DXF Contours

Fundamentals for DXF import

Contours that exist in DXF format can be imported into TURN PLUS.

DXF contours describe

- Workpiece blanks
- Finished parts
- Contour trains
- Milling contours

For workpiece blank and finished-part contours as well as for contour trains, DXF layers should contain only one contour. Milling contours can contain and import multiple contours.

Requirements of a DXF contour or DXF file:

- Only two-dimensional elements
- The contour must be in a separate layer (without dimension lines, without wraparound edges, etc.)
- Turning contours (workpiece blanks or finished parts) should be shown above the turning center (if this is not the case, then they must be revised in TURN PLUS)
- No complete circles, no splines, no DXF blocks (macros), etc.
- The imported contours may consist of no more than 4000 elements (lines, arcs). In addition, up to 10 000 polyline points are permitted.
- The file name can have up to 20 characters

Contour preparation: Since the DXF format is fundamentally different from the TURN PLUS format, the contour is converted from DXF to TURN PLUS format during the import. The following changes/ additions are made during this process:

- Gaps between contour elements are closed
- Polylines are transformed into linear elements
- The starting point of the contour is defined
- The direction of rotation of the contour is defined

Sequence of a DXF importation:

- ▶ Selection of the DXF file
- ▶ Selection of the layer, which contains only the contour(s)
- ▶ Import of the contour(s)
- ▶ Saving (and editing) of the contour in TURN PLUS

Configuring the DXF Import

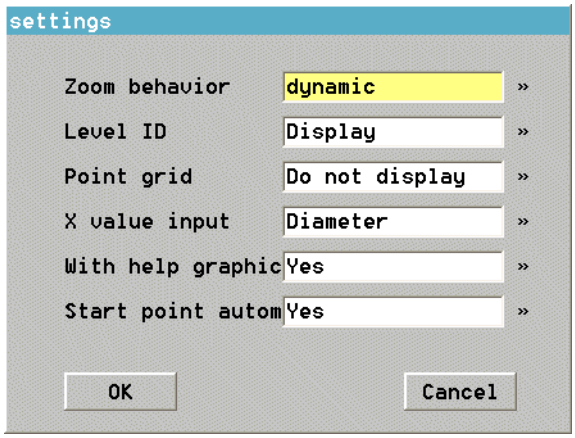
In the **starting point automatic** configuration parameter you specify the behavior of TURN PLUS when entering the finished part contour.

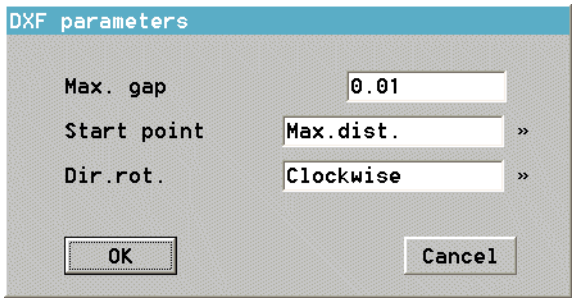
- Select “Configuration > Change > Settings” from the main menu. TURN PLUS opens the “Settings” dialog box.
- Set the “Starting point automatic” parameter:
  - **Yes:** Upon calling the finished part contour entry, TURN PLUS immediately branches to the entry of the contour starting point. The **DXF import** soft key is not available.
  - **No:** After the finished-part contour entry is called, you have the choice of whether a finished-part contour/DXF contour should be imported, or whether the contour will be entered manually.

Only the entry of the finished part contour is affected by this setting. For all other contours you select the form of contour entry either by menu or by soft key.

You can influence the preparation of the contour during the DXF importation with the DXF parameters:

- Select “Configuration > Change > DXF parameters” from the main menu. TURN PLUS opens the “DXF parameters” dialog box.
- Make the required settings.





DXF parameters:

- **Maximum gap:** There might be small gaps between contour elements in the DXF drawing. With this parameter you specify how large the distance between two contour elements may be.
  - **Maximum gap** is not exceeded: The following element is seen as being part of the "current" contour.
  - **Maximum gap** is exceeded: The following element is an element of the "new" contour.
- **Starting point:** The DXF import analyzes the contour and determines the starting point. Possible settings:
  - **Right, left, top, bottom:** The starting point is set to be the contour point that is the furthest to the right (or left, or ...). If more than one contour point satisfies this requirement, then one of these points is selected automatically.
  - **Maximum distance:** The DXF import sets the starting point to one of the two contour points farthest apart from each other. The program automatically determines which of these points is the starting point. It is not possible to influence this decision.
  - **Marked point:** If one of the contour points in the DXF drawing is marked with a complete circle, then this point is specified as the starting point. The contour point must be at the center of the complete circle.
- **Direction of rotation:** Indicate whether the contour is aligned in clockwise or counterclockwise direction.

Save the settings:

- Select "Configuration > Save" in the main menu. TURN PLUS opens the "Save configuration" dialog box.
- Select the "Standard" file and save the changed configuration

DXF import

The **DXF import** function is always offered when a contour entry is necessary. The procedure for DXF importation is the same regardless of the type of contour to be imported (workpiece blank, finished part, etc.).

DXF import:



- Press the soft key: TURN PLUS opens the "DXF import" selection box.
- Choose and load a DXF file.



- Select the contour(s) to be imported.



- The selected contour(s) turn(s) red, and the contour elements of other layers turn yellow.



- Import the DXF contour(s).

6.11 Manipulating Contours

Note when editing contours:

- For contour elements that are superimposed by form elements, the indicated end points or the end points to be entered are given with respect to the "theoretical" end point. When contour elements are modified, the system automatically adjusts chamfers, rounding arcs, threads and undercuts to the new position.
- Sequence, starting point and end point of a contour element are determined by the direction of definition.
- After trimming, deleting or inserting operations have been executed, TURN PLUS analyzes whether successive elements can be combined to form a line segment or an arc. The edited contour is **standardized**.



The turning contour cannot be edited if contours machined with the C or Y axis exist.

Editing the contours of a blank part

If there is a **standard blank** (bar, tube) you can:

Delete:

- Select "Manipulate > Delete > Contour" in the blank part menu. TURN PLUS deletes the blank part.

Resolve:

- Select "Manipulate > Resolve" in the blank part menu. TURN PLUS resolves the standard blank into the individual contour elements. You can then manipulate the individual elements.

If there is a **casting** or if a **blank with individual elements** was defined, manipulate it like a finished part.

Deleting contour elements

Deleting contour elements or form elements:

- Select "Manipulate > Delete > Element (or Form element)" in the finished part menu.
- Select the element to be deleted.
- TURN PLUS deletes the selected contour/form element.

Deleting all form elements:

- Select "Manipulate > Delete > All form elements" in the finished part menu.
- TURN PLUS deletes all existing form elements.

Deleting a finished part contour:

- Select "Manipulate > Delete > Contour" in the finished part menu.
- TURN PLUS deletes the complete finished part contour.

Deleting a C-axis contour:

- Select the front-face, rear-face or lateral-surface window.
- Select "Manipulate > Delete > Pocket/Figure/Pattern" in the finished part menu.
- Select the figure, pattern, etc. to be deleted.
- TURN PLUS deletes the selected contour.

Editing contour elements or form elements

When editing contour elements, TURN PLUS differentiates between:

- "Change > Contour element": You edit the contour element and TURN PLUS adjusts the following elements to the changes.
- "Change > Contour element with displacement": You edit the contour element and TURN PLUS shifts the following elements.

Editing a contour element:

- Select "Manipulate > Change > Form element" (or ".. > Contour element with displacement") in the finished part menu.
- Select the element you want to change. TURN PLUS opens the corresponding line/arc dialog box for editing.
- Change parameters.
- TURN PLUS depicts the modified contour. If more than one solution is displayed, select the most suitable one.
- You can apply ("Confirm" soft key) or cancel (ESC key) the changes.

Editing a form element:

- Select “Manipulate > Change > Form element” in the finished part menu.
- Select the form element you want to edit. TURN PLUS opens the corresponding dialog box for editing.
- Change parameters.
- TURN PLUS applies the changes.

Editing a C-axis contour:

- Select the front-face, rear-face or lateral-surface window.
- Select “Manipulate > Change > Pattern/Figure/Pocket” in the finished part menu.
- Select the figure, pattern, contour element, etc. to be edited. TURN PLUS opens the corresponding dialog box for editing.
- Change parameters.
- TURN PLUS immediately applies the changes for figures. For “free contours,” TURN PLUS depicts the modified contour. You can apply (“Confirm” soft key) or cancel (ESC key) the changes.

Adding a contour or contour element

You can add an individual contour element or a contour (more than one contour element) to an existing contour.

Adding a contour element:

- Select “Manipulate > Insert > Line (or “ ..> Arc”) in the finished part menu.
- Select “Select point.” (The element is inserted after the selected contour element.)
- Select the direction of the line or the direction of rotation of the arc. TURN PLUS opens the corresponding dialog box.
- Define the contour element.
- TURN PLUS integrates the contour element and adjusts the existing contour.

Adding more than one contour element:

- Select “Manipulate > Insert > Contour” in the finished part menu.
- Select “Select point.” (The element is inserted after the selected contour element.)
- Enter the new contour element for element.
- TURN PLUS integrates the new contour and adjusts the existing contour.

Closing the contour

Close an open contour:

- Select “Manipulate > Connect” in the finished part menu.
- TURN PLUS closes the contour by inserting a line segment.

Resolving a contour

With the “Resolve” function, TURN PLUS transforms form elements, figures or patterns into separate contour elements.

- Turning contour: Form elements (also chamfers and rounding arcs) are transformed into linear segments and arcs.
- Contours on the front/rear face or lateral surface: Figures and patterns are resolved into line segments and arcs.

Resolve contour:

- Select “Manipulate > Resolve” in the finished part menu.
- Select the form element, figure or pattern.
- TURN PLUS resolves form elements, figures or patterns into separate contour elements.

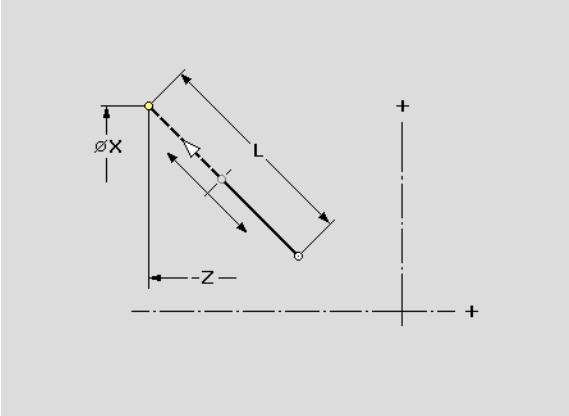


Form elements/figures/patterns once resolved cannot be connected again.

Trimming – Linear element

With this function you change the length of a linear element. The starting point of the contour element remains unchanged.

- Closed contours: The manipulated element is recalculated and the position of the subsequent element is adapted.
- Open contours: The manipulated element is recalculated and the subsequent contour train is shifted.



Parameters

L Length of the modified linear element

X End point of the modified linear element

Z End point of the modified linear element

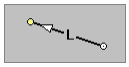
Successor:

- Change angle to following element
- Do not change angle to following element

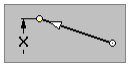
Change the length of a linear element:

▶ Select “Manipulate > Trimming > Length element” in the finished part menu.

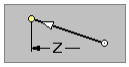
▶ Select the element you want to change. TURN PLUS opens the “Change line length” dialog box.



▶ Enter the new length or



▶ Enter the new end point in X or



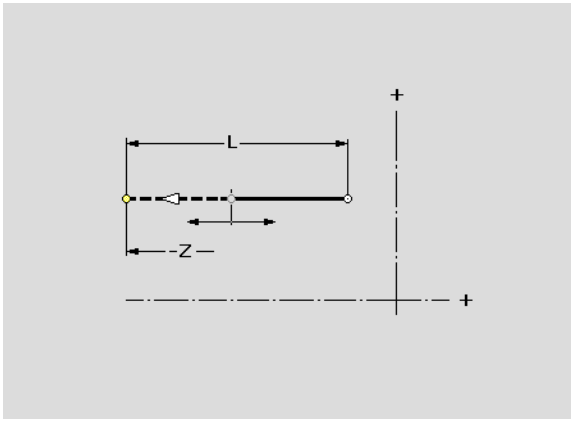
▶ Enter the new end point in Z.

▶ Choose the required setting in the “Successor” input box (with/without angle change).

▶ TURN PLUS integrates the changes and depicts the manipulated contour. You can apply (“Confirm” soft key) or cancel (ESC key) the changes.

Trimming – Length of contour

With this function you change the length of a contour. Select the element to be modified and a "compensation element."



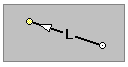
Parameters

L Length or end point of the modified linear element.

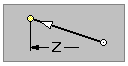
Z Length or end point of the modified linear element.

Change length of contour:

- Select "Manipulate > Trimming > Length contour" in the finished part menu.
- Select the element you want to change. TURN PLUS suggests a compensation element.
- Select the compensation element. TURN PLUS opens the "Change line length" dialog box.



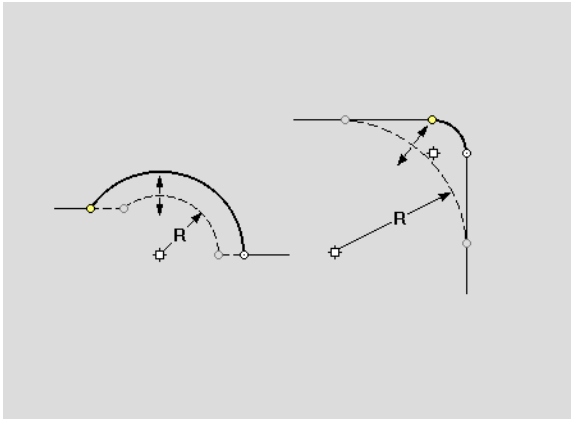
- Enter the new length or



- Enter the new end point in Z.
- TURN PLUS integrates the changes and depicts the manipulated contour. You can apply ("Confirm" soft key) or cancel (ESC key) the changes.

Trimming – Radius of arc

With this function you change the radius of a circular arc.



Parameters

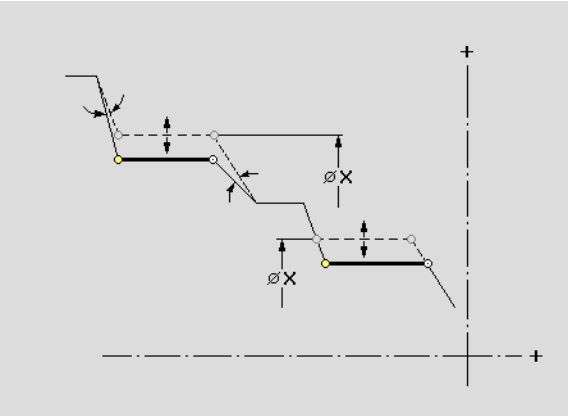
R Radius

Change the radius of the arc:

- Select "Manipulate > Trimming > Radius" in the finished part menu.
- Select the element you want to change. TURN PLUS opens the "Change radius" dialog box.
- Enter the new radius. TURN PLUS integrates the changes and depicts the manipulated contour. You can apply ("Confirm" soft key) or cancel (ESC key) the changes.

Trimming – Diameter of linear element

With this function you change the diameter of a horizontal linear element. TURN PLUS recalculates the manipulated element and adjusts the position of the preceding/succeeding element.



Parameters

D New diameter

Predecessor:

- Change angle to previous element
- Do not change angle to previous element

Successor:

- Change angle to following element
- Do not change angle to following element

Change the diameter of a linear element:

- Select “Manipulate > Trimming > Diameter” in the finished part menu.
- Select the element you want to change. TURN PLUS opens the “Change diameter” dialog box.
- Enter the new diameter and select the type of adjustment for the preceding/succeeding element. TURN PLUS integrates the changes and depicts the manipulated contour. You can apply (“Confirm” soft key) or cancel (ESC key) the changes.

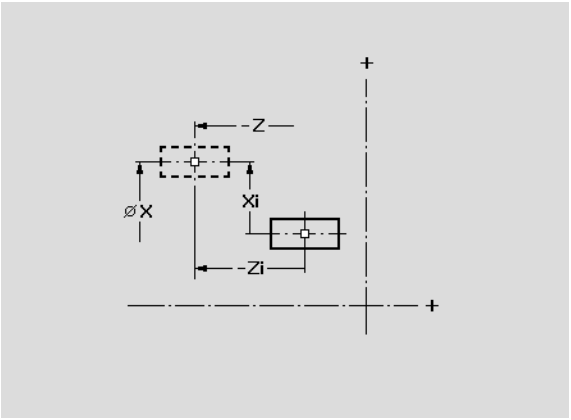
Transformations – Fundamentals

The transformation functions are used for turning contours, contours on the front/rear face or lateral surface, etc.

- Turning contour: The original contour is deleted and the complete turning contour is “transformed.”
- Contours on the front/rear face or lateral surface: Define whether you wish to delete the original contour or copy and “transform” it.

Transformations – Shifting

This function shifts the contour incrementally or to the given position (reference point: contour starting point).



Parameters

X Target point

Z Target point

Xi Target point – incremental

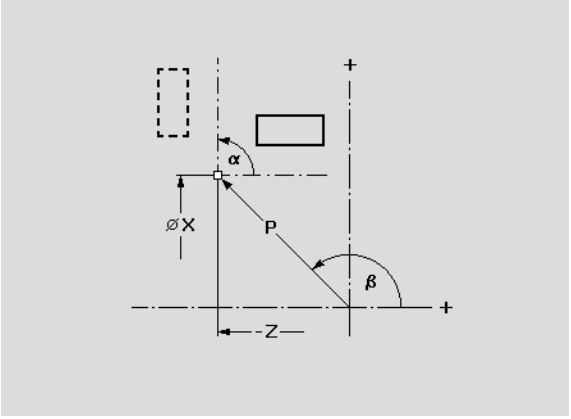
Zi Target point – incremental

Original (only with C-axis contours):

- Copy: Original contour is maintained
- Delete: Original contour is deleted

Transformations – Rotating

This function rotates the contour at the **center of rotation** about the **angle of rotation**.



Parameters

X Center of rotation in Cartesian coordinates

Z Center of rotation in Cartesian coordinates

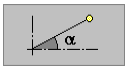
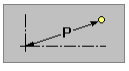
a Center of rotation in polar coordinates

P Center of rotation in polar coordinates

W Angle of rotation

Original (only with C-axis contours):

- Copy: Original contour is maintained
- Delete: Original contour is deleted

Soft keys	
	Polar dimension of the center of rotation: Angle a
	Polar dimension of the center of rotation: Radius

Transformations – Mirroring

This function mirrors the contour. Define the position of the **mirror axis** by entering the starting point and end point or the starting point and angle.

Parameters

X Starting point in Cartesian coordinates

Z Starting point in Cartesian coordinates

XE End point in Cartesian coordinates

ZE End point in Cartesian coordinates

W Rotary angle

a Starting point in polar coordinates

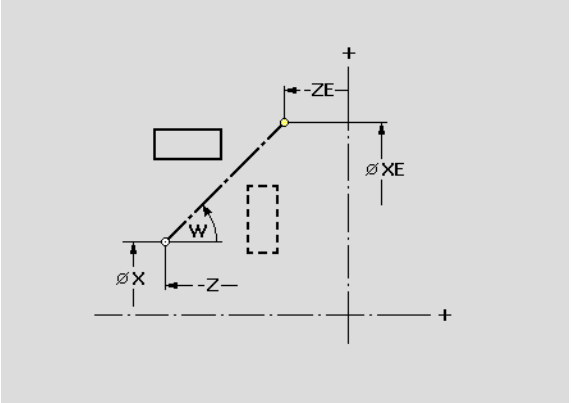
P Starting point in polar coordinates

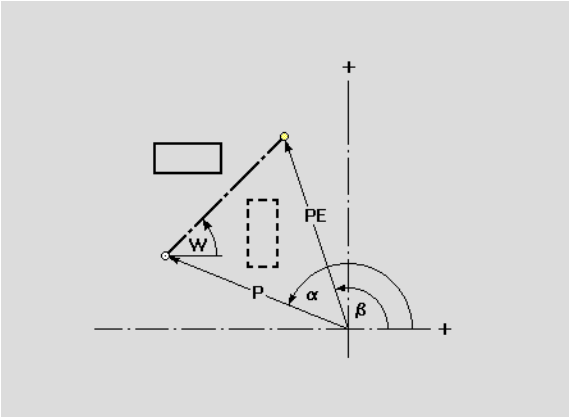
b End point in polar coordinates

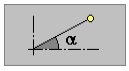
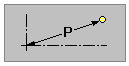
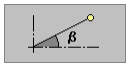
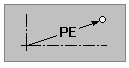
PE End point in polar coordinates

Original (only with C-axis contours):

- Copy: Original contour is maintained
- Delete: Original contour is deleted





Soft keys for polar dimensions	
	Dimension of the center of rotation: Angle a
	Dimension of the center of rotation: Radius
	Dimension of the end point: Angle b
	Dimension of the end point: Radius

Transformations – Inverting

This function inverts the definition direction of the contour.

6.12 Assigning Attributes

After the contour of a blank/finished part has been defined, individual contour elements/contour sections can be assigned attributes. The AWG and IWG evaluate the attributes for generating a working plan.

The machining attributes you define are transferred as cycle parameters by the IWG.

Attributes for workpiece blanks

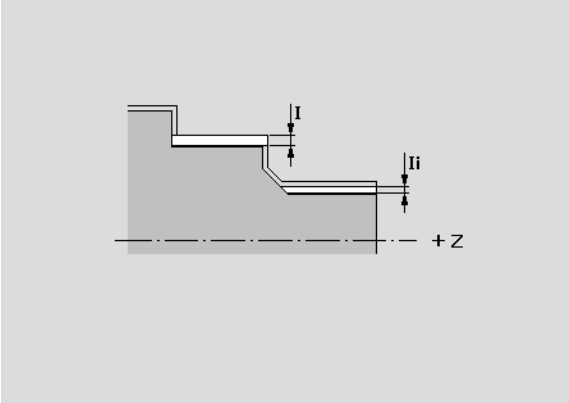
Attributes for workpiece blanks influence the division into machining areas and the selection of roughing cycles in the AWG.

Assign attributes to blanks:

- Select "Workpiece > Blank > Attribute." TURN PLUS opens the "Surface quality" dialog box.
- Define the "Type semifinished:"
  - **Cast or forged workpiece blank:** Working plan generation according to the "cast machining" strategy (first transversal, then longitudinal roughing).
  - **Preturned workpiece blank:** Working plan generation according to the standard strategy. In addition to standard machining cycles, contour-parallel roughing cycles are used.
  - **"Unknown" (or no attribute defined):** Working plan generation according to the standard strategy.

Attributes – Oversize

The attribute defines oversizes for individual contour sections or for the entire contour. The oversize, e.g. grinding allowance, is retained after the machining process has been completed.



Parameters

I Absolute oversize

Ii Relative oversize

TURN PLUS differentiates between:

- **Absolute oversize:** Is "final," other oversizes are ignored.
- **Relative oversize:** Applies additively to other oversizes.

Define the "Oversize" attribute:

- Select "Attribute > Oversize" in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS opens the "Oversize" dialog box.

+

- With the "Continue" key, choose whether you want to enter an absolute or incremental oversize.
- Enter the oversize.

Attributes – Feed rate

The "Feed rate" and "Feed rate reduction" attributes influence the finishing feed rate.

Parameters

F (Finishing) feed rate

Assign the feed-rate attribute:

- Select "Attribute > Feed/Peak-to-valley > Feed rate" in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS opens the "Feed rate" dialog box.
- Define the feed rate. The input value is considered the finishing feed rate.

Parameters

E Factor (finishing feed rate = current feed rate * E)

Assign the feed-rate reduction attribute:

- Select "Attribute > Feed/Peak-to-valley > Feed rate reduction" in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS opens the "Feed rate reduction" dialog box.
- Define the feed rate reduction. The input value is multiplied by the current feed rate.

Attributes – Peak-to-valley

The peak-to-valley attribute is evaluated in finishing cycles. TURN PLUS differentiates between:

- Peak-to-valley (profile depth) (Rt)
- Mean peak-to-valley (Ra)
- Mean peak-to-valley height (Rz)

Parameters

Rt Peak-to-valley (profile depth)

Ra Mean peak-to-valley

Rz Mean peak-to-valley height

Assign the peak-to-valley attribute:

- Select "Attribute > Feed/Peak-to-valley > Peak-to-valley Rt" (or "Mean peak-to-valley Ra" or "Mean peak-to-valley height Rz") in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS opens the corresponding dialog box.
- Define the peak-to-valley specifications.

Attributes – Additive compensation

With this attribute, you can assign an additive compensation value to the entire contour, a contour section or individual contour elements.

The CNC PILOT manages 16 tool-independent additive compensation values. In the attribute, you enter the number of the additive compensation value. The compensation value is defined by parameter.

Parameters

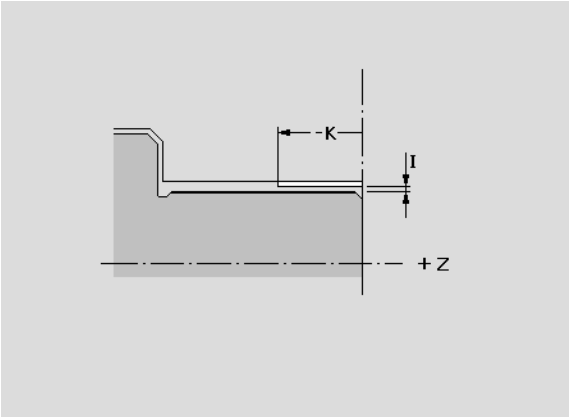
D9xx	Offset, number of the additive compensation value (1 to 16)

Assign the additive compensation:

- Select "Attribute > Feed/Peak-to-valley > Additive compensation" in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS opens the "Additive compensation" dialog box.
- Define the number of the additive compensation.

Machining attributes – Measure

The machining attribute integrates the **expert program** entered in machining parameter 21 ("SP-MEAS01"). You use it to organize a measuring cut after every nth workpiece.



Parameters

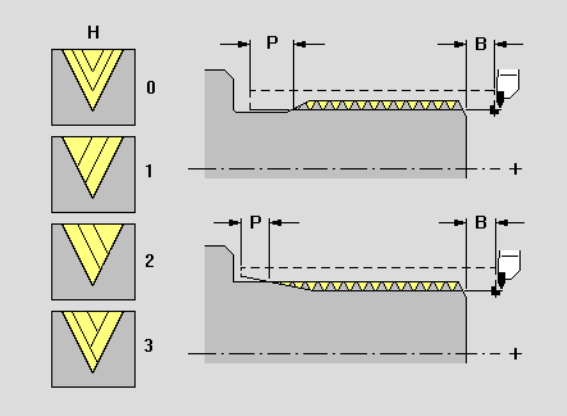
I	Oversize for measuring cut
K	Length of measuring cut
Q	Measuring loop counter, every nth workpiece is measured

Assign the "Measure" machining attribute:

- Select "Attribute > Machining attribute > Measure" in the finished part menu.
- Select the contour element. TURN PLUS opens the "Measuring cut" dialog box.
- Enter the parameters of the expert program.

Machining attributes – Threading

The machining attribute defines the details of a thread cutting operation.



Parameters

B Starting length

- No input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses.
- No input, no undercut/recess: The CNC PILOT uses the thread starting length from machining parameter 7.

P Overrun length

- No input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses.
- No input, no undercut/recess: The CNC PILOT uses the thread run-out length from machining parameter 7.

C Starting angle, if the beginning of the thread is defined with respect to rotationally nonsymmetrical contour elements

I Maximum infeed

V Type of infeed

- V=0: Constant cross section for all cuts
- V=1: Constant infeed
- V=2 (distribution of remaining cuts): If the division thread depth/infeed provides a remainder, the first infeed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
- V=3 (EPL method): Infeed is calculated from the pitch and spindle speed

H Type of offset used for the individual infeeds for smoothing the thread sides

- H=0: Without offset
- H=1: Offset from left
- H=2: Offset from right
- H=3: Tool is offset alternately from the right and left

Q Number of air cuts after the last cut (for reducing the cutting pressure in the thread base)

Assign the “Threading” machining attribute:

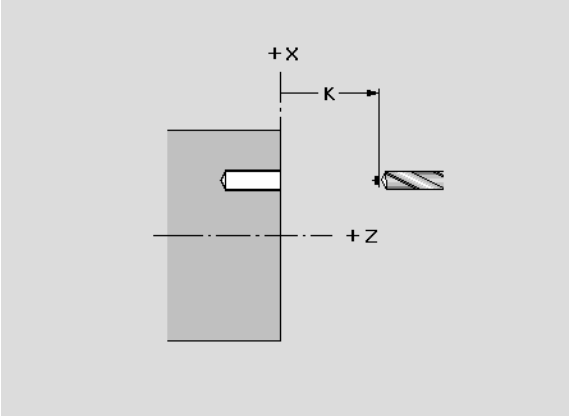
- ▶ Select “Attribute > Machining attribute > Threading” in the finished part menu.
- ▶ Select the thread. TURN PLUS opens the “Threading” dialog box.
- ▶ Enter the thread parameters.

Machining attributes – Drill – Retraction plane

The machining attribute defines the retraction plane of a drilling operation. The drill moves to the retraction plane (hole on lateral surface: diameter) before/after the drilling operation.

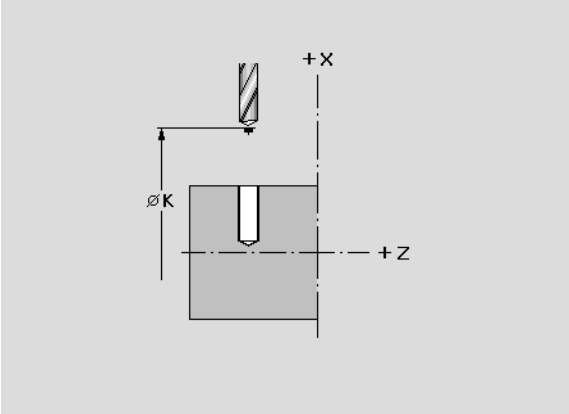
Parameters

K Retraction plane. Position of the drill before/after machining.



Assign the "Retraction plane" machining attribute:

- Select "Attribute > Machining attribute > Drill > Retraction plane" in the finished part menu.
- Select the bore hole. TURN PLUS opens the "Retraction plane drilling" dialog box.
- Define the retraction plane.



Machining attributes – Drilling combinations

The machining attribute influences the tool selection. TURN PLUS supports the following tool combinations:

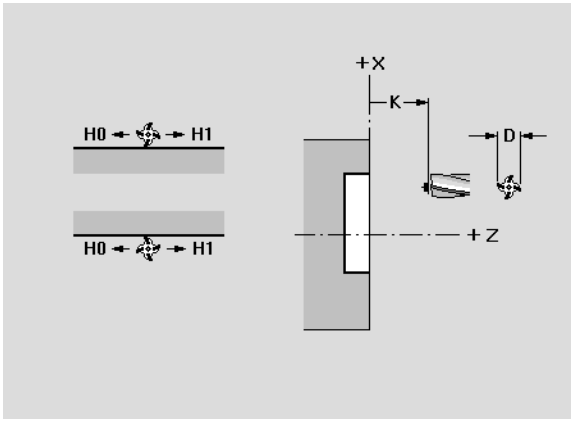
- **Centering and countersinking:** NC centering drill (type 32*); alternative tool: Centering drill (type 31*)
- **Drilling and countersinking:** Step drill (type 42*)
- **Drilling with thread:** Tap drill (type 44*)
- **Drilling with reaming:** Delta drill (type 47*)

Assign the "Drilling combinations" machining attribute:

- Select "Attribute > Machining attribute > Drill > Centering and countersinking (or Drilling and countersinking, Drilling with thread, Drilling with reaming)" in the finished part menu.
- Select the bore hole.
- TURN PLUS assigns the machining attribute.

Machining attributes – Contour milling

The attribute defines the contour milling operation and the associated machining parameters for the selected figure or for a "free" open or closed contour.



Parameters

Q Cutter position

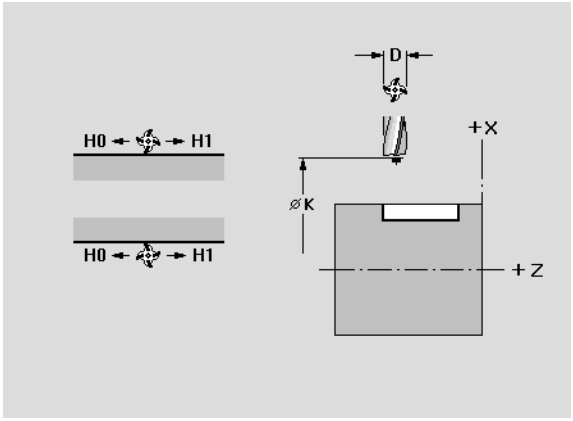
- Contour: Milling center on the contour
- For closed contours:
  - Inside (milling)
  - Outside (milling)
- For open contours:
  - Left of the contour (in machining direction)
  - Right of the contour (in machining direction)

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

D Milling diameter for selecting suitable tool.

K Retraction plane. Cutter position before/after milling (lateral surface: diameter).

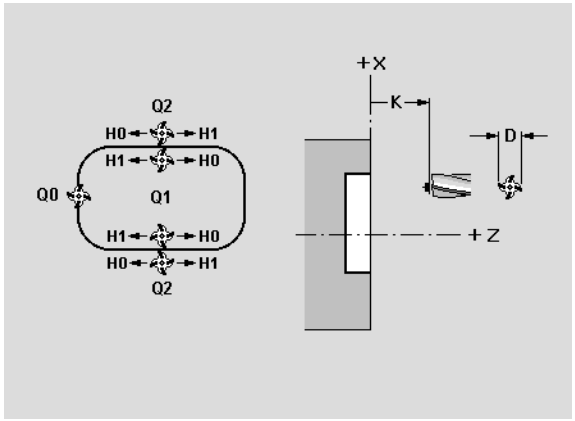


Assign the "Contour milling" machining attribute:

- Select "Attribute > Machining attribute > Mill > Contour milling" in the finished part menu.
- Select the contour to be milled. TURN PLUS opens the "Contour milling" dialog box.
- Define the milling parameters.

Machining attributes – Area milling

The attribute defines the area milling operation and the associated machining parameters for the selected figure or for a "free" closed contour.



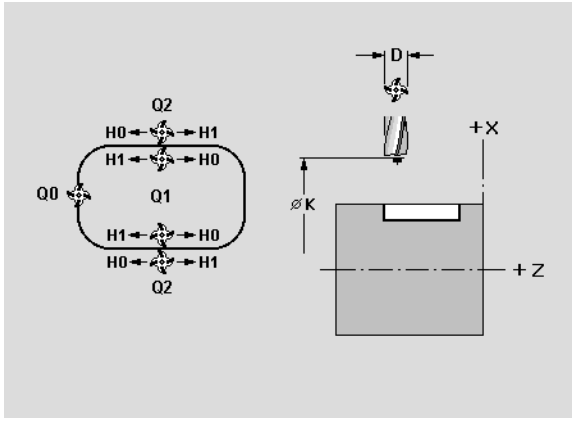
Parameters

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

D Milling diameter for selecting suitable tool.

K Retraction plane. Cutter position before/after milling (lateral surface: diameter).

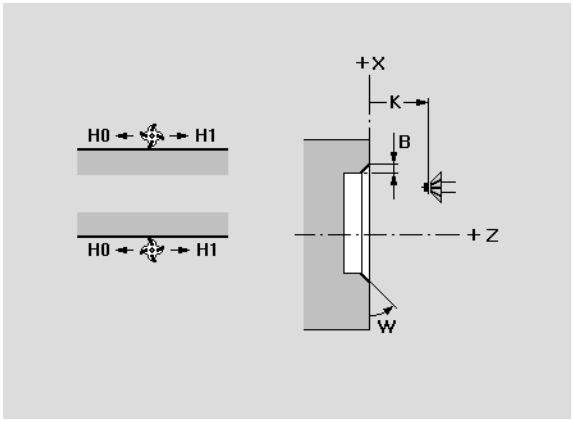


Assign the "Area milling" machining attribute:

- Select "Attribute > Machining attribute > Mill > Area milling" in the finished part menu.
- Select the contour to be milled. TURN PLUS opens the "Area milling" dialog box.
- Define the milling parameters.

Machining attributes – Deburring

The attribute defines the deburring operation and the associated machining parameters for the selected figure or for a "free" open or closed contour.



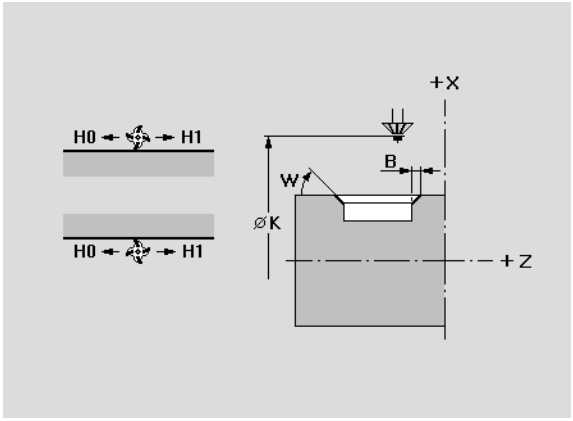
Parameters

H Cutting direction
- 0: Up-cut milling
- 1: Climb milling

B Width

W Angle for the tool selection (default 45°)

K Retraction plane. Cutter position before/after milling (lateral surface: diameter).

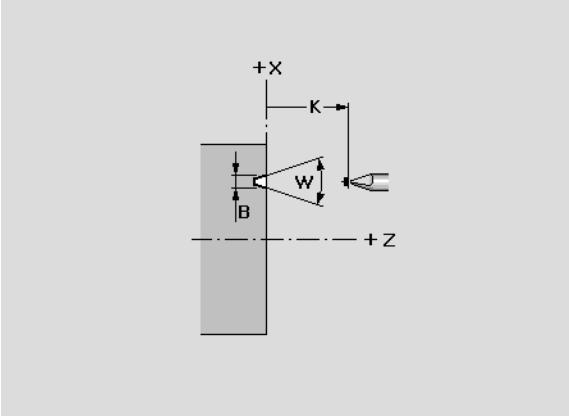


Assign the "Deburring" machining attribute:

- Select "Attribute > Machining attribute > Mill > Deburring" in the finished part menu.
- Select the contour to be milled. TURN PLUS opens the "Deburring" dialog box.
- Define the milling parameters.

Machining attributes – Engraving

The attribute defines the engraving operation and the associated machining parameters for the selected figure or for a "free" open or closed contour.



Parameters

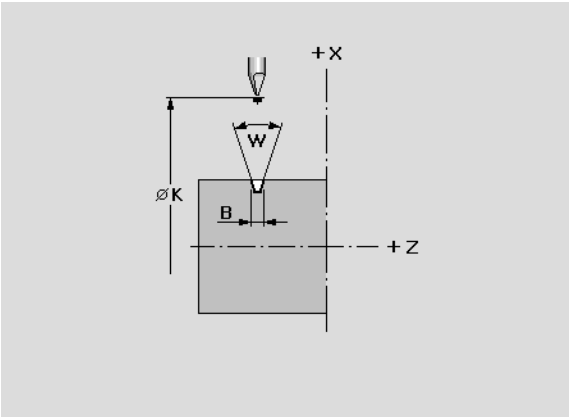
B Width

W Angle for the tool selection (default 45°)

K Retraction plane. Cutter position before/after milling (lateral surface: diameter).

Assign the "Engraving" machining attribute:

- Select "Attribute > Machining attribute > Mill > Engraving" in the finished part menu.
- Select the contour to be milled. TURN PLUS opens the "Engraving" dialog box.
- Define the milling parameters.



Machining attributes – Precision stop

The attribute defines a precision stop for the selected contour elements or contour sections.

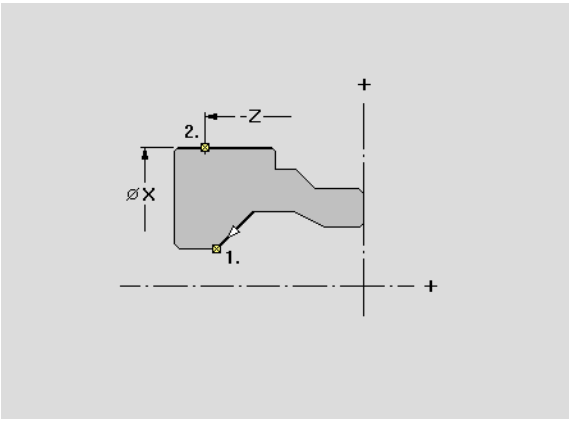
Assign the precision stop:

- Select "Attribute > Precision stop" in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see "Selections" on page 448).
- TURN PLUS assigns the machining attribute.

Machining attributes – Separation point

The attribute defines a position on the contour as the separation point.

Separation points are used for shaft machining and for machining with more than one setup.



Parameters

Position

- Delete: Deletes the existing separation point. The separation of the contour elements is maintained.
- 1. At target point: Separation point is end point of element
- 2. On element: Separation point is on the element.

X X position of the separation point

Z Z position of the separation point

Assign the separation point:

- Select "Attribute > Separation point" in the finished part menu.
- Select the contour element. TURN PLUS opens the "Separation point" dialog box.
- Define the exact position of the separation point (end point of the element or position on the element). Alternatively, delete a defined separation point.

Attributes – Exclusion from machining

The "Exclusion from machining" attribute is evaluated by the AWG. The effect depends on the type of machining:

- **Roughing:** The attribute is evaluated with the first/last element of an inside/outside contour. Form elements are not machined.
- **Finishing:** Marked elements are not finished.
- **Predrilling:** The attribute is ignored.
- **Recessing:** Marked recesses are not machined.
- **Thread machining:** Marked thread elements are not finished and the thread is not cut.
- **Centering:** Marked holes (form elements) are not drilled.
- **Drilling:** Marked holes for C/Y machining are not machined.
- **Milling:** Marked milling contours for C/Y machining are not machined.

Assigning the “Exclusion from machining” attribute to the elements of the turning contour:

- Select “Attribute > Feed/Peak-to-valley > Exclusion from machining” in the finished part menu.
- Select the entire contour, a contour section or individual contour elements (see “Selections” on page 448).
- TURN PLUS assigns the attribute.

Assigning the “Exclusion from machining” attribute to a C/Y-axis contour:

- Activate the front-face, rear-face or lateral-surface window.
- Select “Attribute > Machining attribute > Drill (or Mill) > Exclusion from machining” in the finished part menu.
- Select the drilling or milling contour.
- TURN PLUS assigns the attribute.

Deleting machining attributes

You can delete the machining attributes of holes and milling contours.

Deleting the “Drill” machining attribute:

- Select “Attribute > Machining attribute > Drill > Delete drilling attributes” in the finished part menu.
- Select the bore hole.
- TURN PLUS deletes the machining attributes of this hole.

Deleting the “Mill” machining attribute:

- Select “Attribute > Machining attribute > Mill > Delete milling attributes” in the finished part menu.
- Select the milling contour.
- TURN PLUS deletes the machining attributes of this milling contour.

6.13 Preparing a Machining Process

Preparing a machining process – Fundamentals

The “Prepare” function defines the chucking equipment, the position of the chucking equipment, and TURN PLUS specific turret assignments.

For chucking a workpiece, TURN PLUS determines the following data:

- Inside and outside cutting limit.
- Zero point shift. The value is transferred to the NC program as a G59 command.

TURN PLUS transfers the following set-up information to the program head:

- Clamping diameter
- Free length
- Clamping pressure



- You can set/change the cutting limits.
- If you do not use “clamp” TURN PLUS assumes standard values.
- You define the chucking equipment for the second setup **after** machining the first setup.
- If you clamp the workpiece at the spindle and the tailstock, TURN PLUS assumes that a **shaft is machined** (see “Shaft machining” on page 557).

Chucking a workpiece at the spindle

Clamp the workpiece:

- Select “Prepare > Chucking > Clamp > Spindle side.”
- Select the type of chuck (drop-down menu). TURN PLUS opens one of the following dialog boxes:
  - Two-jaw chuck
  - Three-jaw chuck
  - Four-jaw chuck
  - Collet chuck
  - Without chuck (face driver)
  - Three-jaw chuck indirect (face driver with jaws)
- Define the chuck and jaws as well as the clamping form and clamp range.
- TURN PLUS displays the selected chucking equipment and depicts the cutting limit as a red line.

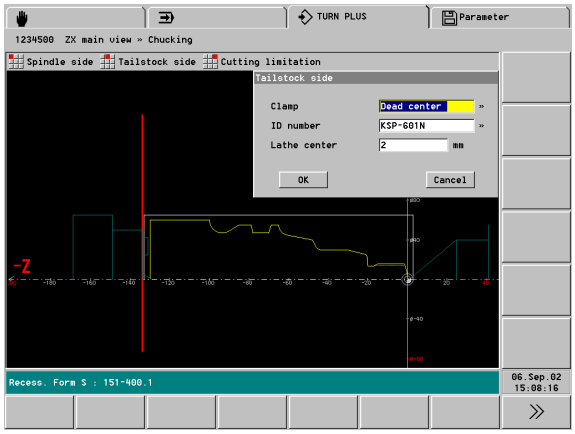


Select first the type of chuck and the jaw type. TURN PLUS takes these data into account in the selection of the ID number of the chuck/jaw.

Chucking a workpiece at the tailstock

Clamp the workpiece:

- Select “Prepare > Chucking > Clamp > Tailstock side.” TURN PLUS opens the “Tailstock side” dialog box.
- Define the chucking equipment at the tailstock.



Parameters

Clamping

Select the chuck type:

- Dead center
- Lathe center
- Centering cone

ID number of the chucking equipment

Lathe center

Depth to which the clamping equipment presses into the material. TURN PLUS uses this value to position the chucking equipment graphics.



If you clamp the workpiece at the spindle and the tailstock, TURN PLUS assumes that a **shaft is machined**.

Defining the cutting limit

TURN PLUS finds the cutting limit on the outside and inside of the contour when the workpiece is chucked at the spindle.

Edit the cutting limit:

- Select "Prepare > Chucking > Clamp > Cutting limitation." TURN PLUS opens the "Cutting limitation for AWG" dialog box.
- Define the cutting limit.

The cutting limit is displayed as a red line.

Parameters

Outside contour

Position of the cutting limit

Inside contour

Position of the cutting limit

Deleting the chucking data

This function deletes all data used for clamping a workpiece and deletes the defined cutting limits.

Delete the chucking data:

- Select "Chucking > Delete chuck plan."

Rechuck – Standard machining

Use "Rechuck – Standard machining" for front-face and rear-face machining with separate NC programs.

TURN PLUS

- Mirrors the workpiece (blank and finished part) and shifts the zero point by "Nvz."
- Rotates lateral surface contours or contours of the YZ plane by "Wvc."
- Deletes the chucking equipment used for the first setup.

Rechuck:

- ▶ Select "Prepare > Chucking > Rechuck > Standard machining." TURN PLUS opens the "Rechuck workpiece" dialog box.
- ▶ Enter the rechucking parameters.

Parameters

Nvz	Zero point shift (proposed value: length of the finished contour)
Wvc	Angular shift



- Save the working plan of the first setup before rechucking. When you use the rechucking function, TURN PLUS deletes the previous working plan and the operating resources used.
- Rechucking is no substitute for chucking.

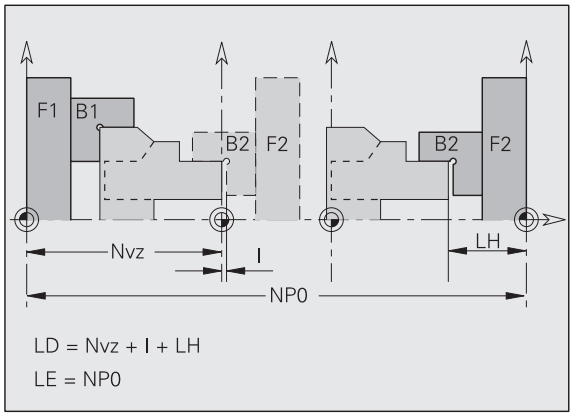
Rechuck – 1st setup after 2nd setup

The "Rechuck – 1st setup after 2nd setup" function initiates the machining operation for the second setup.

First define the chucking equipment. TURN PLUS then activates an **expert program** from machining parameter 21. Which expert program is activated depends on the spindle entries specified for "1st setup .." and "2nd setup .." in the program head as well as on the entry in "Machining sequence:"

- Different spindles entered for 1st setup and 2nd setup (machine with opposing spindle):
  - Main machining and submachining "Rechuck – Full-surface machining:" Entry from "SP-UMKOMPL" (transfer to opposing spindle)
  - Main machining and submachining "Parting – Full-surface machining:" Entry from "SP-UMKOMPLA" (parting and transfer to opposing spindle)
- Same spindle entered for 1st setup and 2nd setup (full-surface machining on machine with one spindle):
  - Main machining and submachining "Rechuck – Full-surface machining:" Entry from "SP-UMHAND" (manual rechucking)
  - Main machining and submachining "Parting – Full-surface machining:" Entry from "SP-ABHAND" (parting and manual rechucking)

The illustration indicates the parameters that are relevant to transferring the workpiece to the opposing spindle.



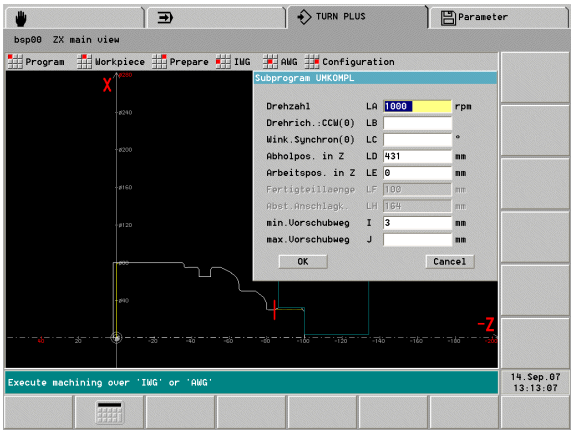
Designations	
F1/B1	Chuck/jaws, main spindle
F2/B2	Chuck/jaws, opposing spindle
Nvz	Zero point shift (G59, ...)
I	Safety clearance on blank part (machining parameter 2)
NP0	Zero point offset (e.g.. MP 1164 for Z axis $1)

The expert programs described on the following pages serve as examples. Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

Expert program "UMKOMPL"

The expert program entered in SP-UMKOMPL (machining parameter 21) transfers the workpiece to the opposing spindle.

TURN PLUS enters the calculated parameters as proposed values. Check, edit or enter the values.



Parameters (example)

LA Spindle speed for workpiece transfer

LB Direction of spindle rotation

- 0: CCW
- 1: CW

LC Speed or angle synchronization

- 0: Angle synchronization without angle offset
- >0: Angle synchronization with predefined angle offset
- <0: Spindle speed synchronization

Parameters (example)

LD Pick-up position in Z

- 0: Pick-up position at machine dimension 1
- 1 to 6: Pick-up position at machine dimension 1 to 6
- ¼ 0 to 6: Pick-up position. TURN PLUS calculates a default value.

LE Working position in Z (proposed value: zero point offset of Z axis $1)

I Minimum feed path

- No traverse to fixed stop: Safety clearance on the workpiece to be transferred (proposed value: "Safety clearance on blank part" from machining parameter 2).
- With traverse to fixed stop: Refer to the machine manual

J Maximum feed path and traverse to fixed stop

- No input: No traverse to fixed stop
- Traverse to fixed stop. For the meaning of the parameter refer to the machine manual.

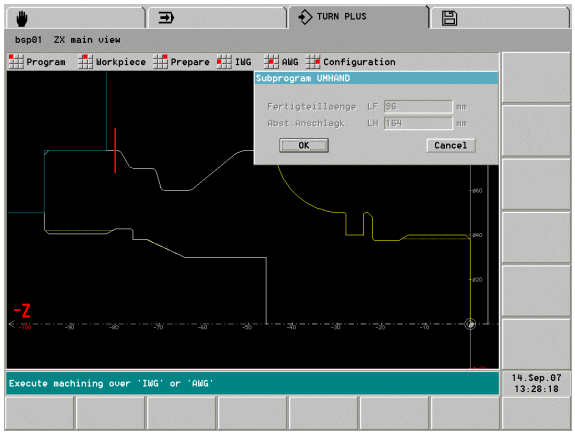
Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

Expert program "UMHAND"

The expert program entered in SP-UMHAND (machining parameter 21) supports the manual rechucking of the workpiece for machining the rear face on machines with one spindle.

TURN PLUS enters the calculated parameters for your information. Check the entries.

Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

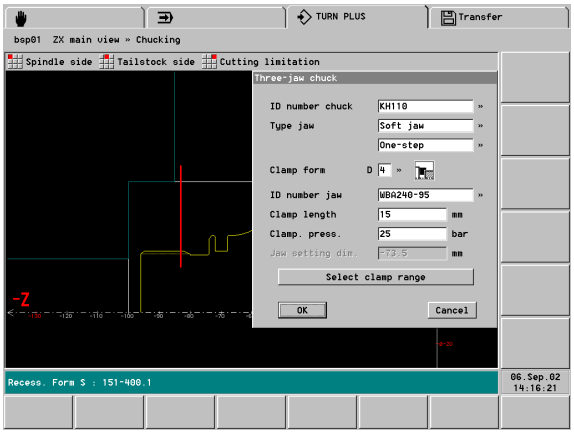


Rechuck – Full-surface machining back to 1st setup

If you wish to correct or optimize the contour or machining process after the 2nd setup has been generated, you can return to the starting point of your machining process:

- Select "Prepare > Chucking > Rechuck > Full-surface machining back to 1st setup." TURN PLUS deletes the work blocks of the second setup.

Parameters for two-jaw, three-jaw or four-jaw chucks



Parameters

ID number chuck

Jaw type and steps

Clamp form (see table below)

ID number jaw

Clamp length

TURN PLUS calculates the clamp length from the jaw and the clamp form. Correct this value if the clamping length is different.

Clamping pressure

The entry is transferred to the program head. TURN PLUS does not evaluate this parameter.

Jaw setting dimension (this entry is for your information)

Distance between outer edge of chuck and outer edge of jaw. Negative dimension: The jaw protrudes from the chuck

"Select clamp range" button

Define the placement of the chucking equipment:

- For contours with chamfers, rounding arcs or arc elements, mark the area around the chuck corner.
- For rectangular parts, mark an adjoining element of the chuck corner.

Clamping form	Unstepped	One-step	Two-step
D=1			
D=2			
D=3			
D=4			
D=5			
D=6			
D=7			

Collet chuck parameters

Parameters

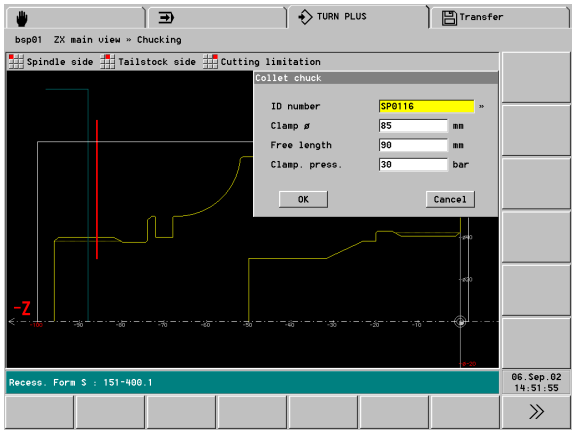
ID number chuck

Clamp diameter

Free length (distance between front edge of collet and right edge of blank)

Clamping pressure

The entry is transferred to the program head. TURN PLUS does not evaluate this parameter.



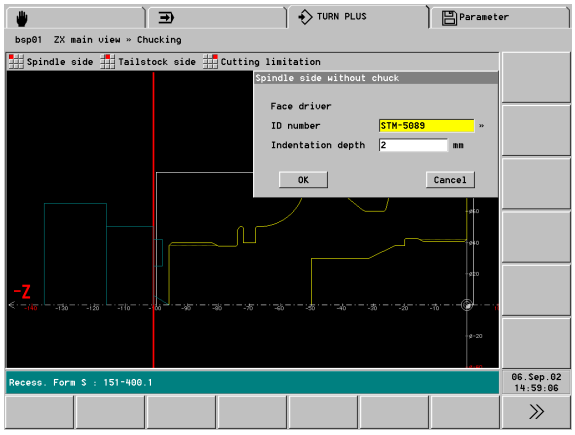
Parameters for face drivers (“without chuck”)

Parameters

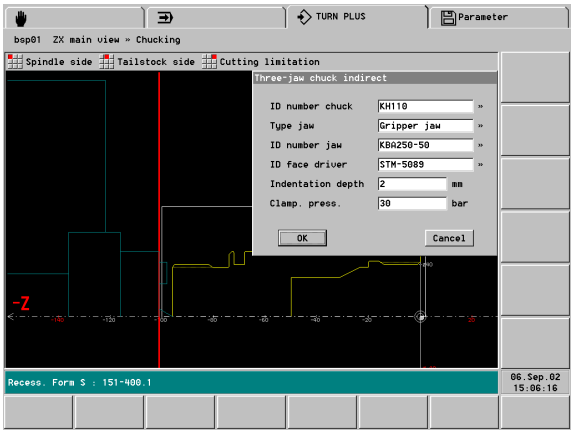
ID number

Indentation depth

Approximate depth to which the grippers press into the material. TURN PLUS uses this value to position the face driver graphics.



Parameters for face drivers with jaws (“Three-jaw chuck indirect”)



Parameters

ID number chuck

Type of jaw

ID number jaw

ID of face driver

Indentation depth

Approximate depth to which the grippers press into the material. TURN PLUS uses this value to position the face driver graphics.

Clamping pressure

The entry is transferred to the program head. TURN PLUS does not evaluate this parameter.

Setting up and managing tool lists

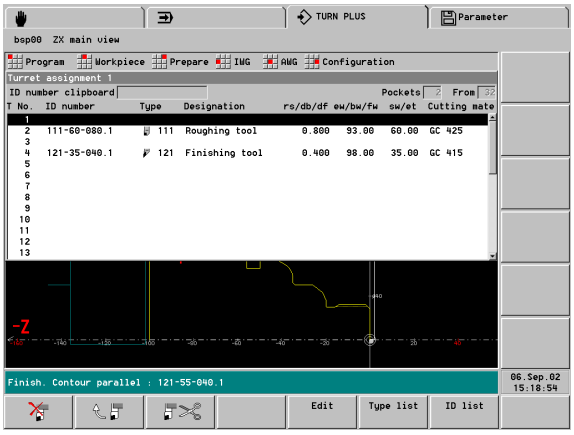
In TURN PLUS, you can define and manage turret assignments as described on the next few pages.



- Load TURN PLUS-specific turret assignments **before** using the IWG/AWG function for selecting tools.
- In the machining parameter 2 “Global technology parameters,” you define the tools to be used by the IWG/AWG.

View the turret assignment:

- Select “Setting up > Tool list > Tool life management.”
- TURN PLUS opens the active tool list.



Setting up tools

Select “Setting up > Tool list > Set up turret > Set up turret n.”

Select the tool location.

Entering the tool directly:

Press ENTER (or the INS key). The CNC PILOT opens the ““Tool” dialog box.

Enter the ID number, define the relevant coolant circuits and close the dialog box.

Select the tool from the database:

Type list — List the tools by type mask or

ID list — List the tools by ID number mask

Place the cursor on the nominal tool.

Insert — Insert the tool.

ESC — Press the ESC key to exit the tool database.



Adjust the coolant circuits in the “Tool” dialog box.

Deleting a tool

Select “Setting up > Tool list > Set up turret > Set up turret n.”

Select the tool location.



Use the soft key, or



Press the DEL key to delete the tool

Changing the tool location

Select “Setting up > Tool list > Set up turret > Set up turret n.”

Select the tool location.



Deletes the tool and saves it in the “ID number clipboard.”

Select a new tool location.



Take the tool from the “ID number clipboard” If the location was occupied, the previous tool is taken into the clipboard.

Managing tool lists

Functions for turret assignment:

- **Load saved tool list:** Loads a saved tool list (“Load file” selection box).
- **Load tool list of machine**: Loads the current turret assignment of the machine.
- **Save list**: Saves the current turret assignment.
- **Delete list:** Deletes the selected file.

Loading a tool list from the file

Select “Prepare > Tool list > Load list > Saved tool list.” TURN PLUS opens the “Load file” selection box.

Select and load the tool list.

Loading the tool list of the machine

Select “Prepare > Tool list > Load list > Tool list of machine.”

TURN PLUS loads the current tool list of this slide.

Saving the tool list

Select “Prepare > Tool list > Save list.” TURN PLUS opens the “Save file” selection box.

Enter a file name and save the tool list.

Deleting a tool list

Select “Prepare > Tool list > Delete list.” TURN PLUS opens the “Delete file” selection box.

Select the file. TURN PLUS deletes the selected tool list.

6.14 Interactive Working Plan Generation (IWG)

In the **IWG**, you define the **work blocks**. You do this by selecting the tool and the cutting values and determining the fixed cycle.

The **semiautomatic mode** of the IWG generates a complete work block.

In **special machining (SM)** you add paths of traverse, subprogram calls or G/M functions (example: use of tool handling systems).

A work block has the following content:

- Tool call
- Cutting values (technology data)
- Approach (may be omitted)
- Machining cycle
- Tool retraction (may be omitted)
- Moving to tool change point (may be omitted)

When the tool data/cutting data of the previous work block are used, TURN PLUS does not generate a new tool call or commands for a new feed rate/spindle speed.

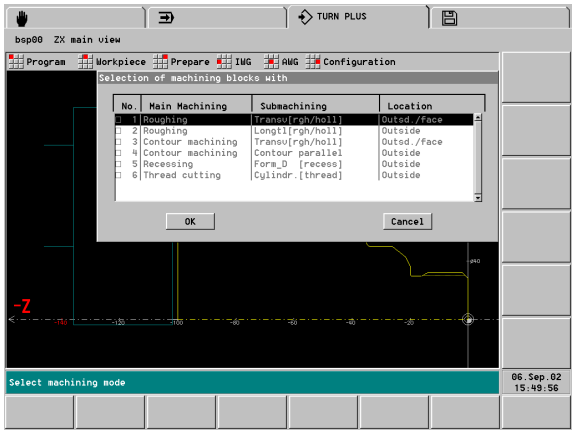
If no working plan exists, TURN PLUS branches directly to the selection of the machining modes. Create the working plan by entering the individual work blocks one after the other.

You can edit an existing working plan and add work blocks.

Working plan exists

If a working plan exists, the IWG starts up with the “Working plan exists” dialog. Set:

- Working plan – new (reject an existing working plan and create a new one)
- Working plan – continue
- Working plan – edit
- Working plan – view



Select IWG. TURN PLUS opens the “Working plan exists” dialog box.

Creating a new working plan:

Select **New**.

TURN PLUS deletes the existing working plan.

Create the working plan by entering work block by work block.

Adding work blocks:

Select **Continue**.

Add further work blocks.

Editing work blocks:

Select “Edit.”

TURN PLUS displays the existing working plan. Select the work blocks you want to modify (see illustration).

TURN PLUS simulates the working plan and stops at the selected work blocks.

Correct/optimize the work block.

Viewing work blocks:

Select “View.”

TURN PLUS displays the existing working plan. Select the work blocks you want to view.

TURN PLUS simulates the working plan and stops at the selected work blocks.

Generating a work block

A work block is defined as follows:

1. Select the machining mode.
2. Select the tool.
3. Check or optimize the cutting data.
4. Define the machining range by area selection (see “Selections” on page 448).
5. Check or optimize the cycle parameters.
6. If required: Define the approach position and/or retraction position.
7. If required: Move to the tool change point.
8. Check the work block with the simulation function.
9. Accept or correct the work block.

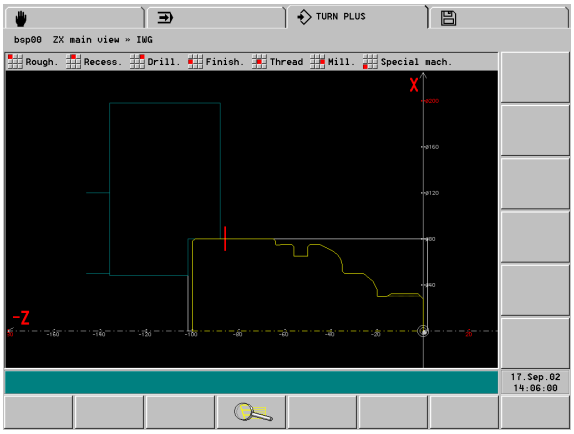
Alternatively, you can start by defining the machining range. TURN PLUS can then select the tool (“Tool > Automatic” menu item).

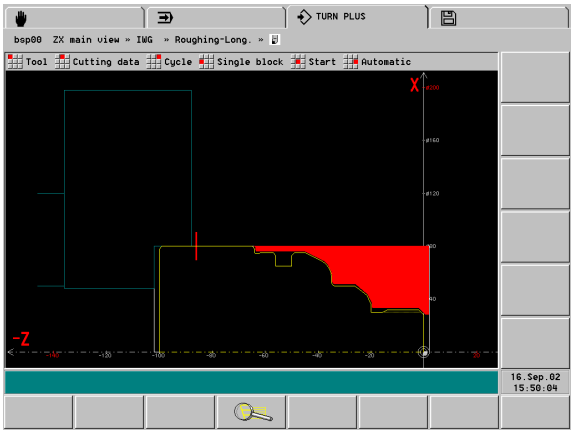
When you have defined all the actions and parameters of the work block, start the simulation (“Start” menu item). After the simulation, you have the following options:

- **Accept the block**: The work block is saved and the workpiece is updated (workpiece blank regeneration).
- **Change the block**: TURN PLUS discards the work block. Correct the parameters and repeat the simulation.
- **Repeat the block**: TURN PLUS simulates the machining process again.

Overview of machining modes:

- Roughing (see “Overview of roughing operations” on page 499)
- Recessing (see “Overview of recessing operations” on page 507)
- Drilling (see “Overview of drilling operations” on page 516)
- Finishing (see “Finishing” on page 521)
- Thread (see “Thread machining (G31)” on page 525)
- Milling (see “Overview of milling operations” on page 526)
- Special machining (see “Special machining (SM)” on page 532)

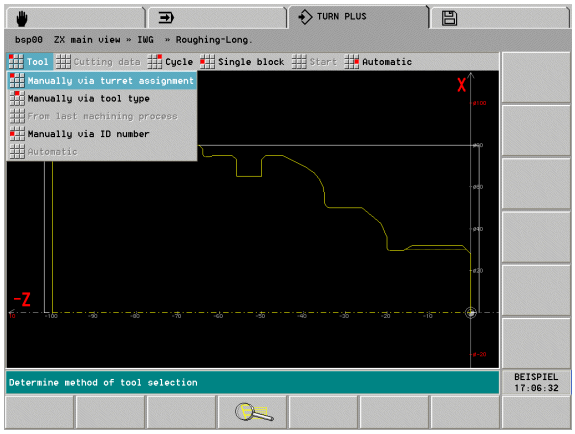




Calling a tool

The “Tool” menu item is only available after you have selected the machining mode. The submenu items have the following meanings:

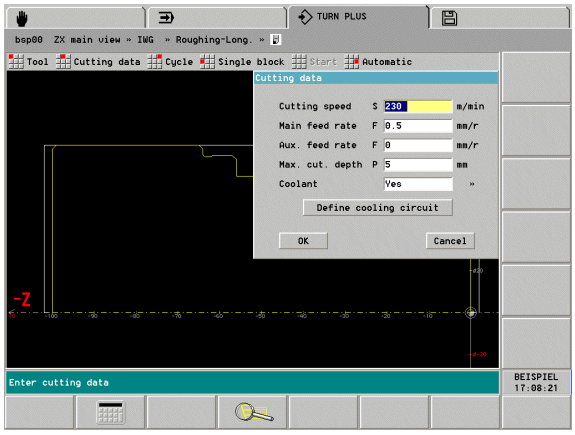
- **Manually via turret assignment:** Select a tool positioned on the turret.
- **Manually via tool type:** Select a tool from the database and position it on the turret.
- **From last machining process:** The IWG selects the tool used for the last machining operation.
- **Manually via tool type/ID number:** Select a tool from the database and position it on the turret.
- **Automatic:** The IWG selects and positions the tool on the turret. – Prerequisite: The machining range has been defined.



Cutting data

After selecting the tool, check/optimize the technology data. Depending on the material to be machined, TURN PLUS determines the cutting data using the tool data (cutting material) from the technology database. Check/optimize the suggested values.

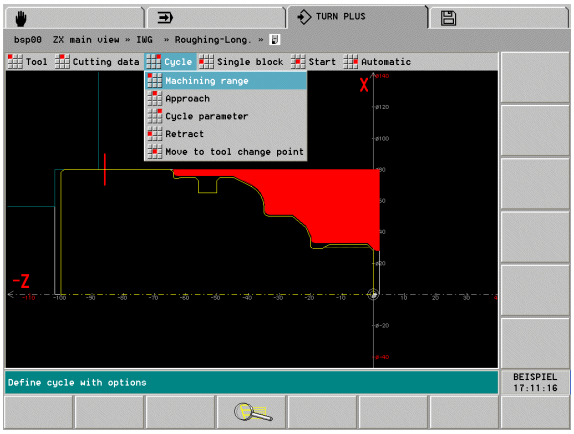
- Cutting speed S
- Main feed rate F
- Auxiliary feed rate F
- Maximum cutting depth P (adopted as cycle parameter)
- Coolant
  - Yes: TURN PLUS generates M commands for switching the coolant circuits ON/OFF.
  - No: TURN PLUS generates no M commands for switching the coolant circuits ON/OFF.
- “Define cooling circuit” button: Opens the “Cooling circuits” dialog box. Define the coolant circuits used.

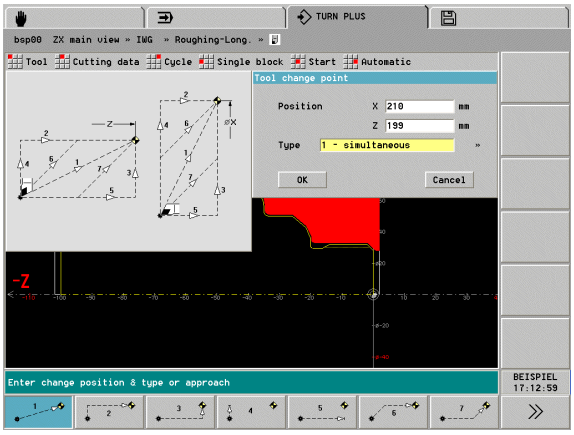


Cycle specification

In the “Cycle” drop-down menu, define the cycle parameters and the strategies for approach and departure:

- **Machining range:** Define the area to be machined and the machining direction by area selection.
  - Selection by soft key: The sequence of selection defines the machining direction.
  - Selection by touch pad – left mouse button: Machine in direction of contour definition.
  - Selection by touch pad – right mouse button: Machine opposite to direction of contour definition.
- **Approach:** Before the cycle is called, the tool moves at rapid traverse from the current position to the approach position. The “Approach” function is not contained in drilling and threading cycles. Place the tool on a suitable position using the “Approach” function.
- **Cycle parameter:** TURN PLUS suggests the cycle parameters. Check/optimize the parameters.
- **Retract:** After the cycle is finished, the tool moves at rapid traverse to the retraction position.
- **Move to tool change point:** After the cycle is finished or after “Retract,” the tool moves to the change position. You define the position to be approached and the traversing mode in “Traversing mode to tool change position [WP]” (machining parameter 2):
  - WP=1: The position defined in the “Tool change point” dialog box is approached with G0. TURN PLUS enters the tool change point as a proposed value.
  - WP=2: TURN PLUS generates a G14. The position defined in the “Tool change point” dialog box has no effect.
  - WP=3: TURN PLUS calculates the tool change point according to the tools currently on the turret.





Danger of collision!

As frequently not all of the tools are known when you generate a work block, you should not use the WP=3 setting (machining parameter 2) in the IWG.

Overview of roughing operations

The IWG presents the following roughing operations for selection ("Roughing" drop-down menu):

- Longitudinal roughing: see "Roughing longitudinal (G810)" on page 500
- Transverse roughing: see "Roughing transverse (G820)" on page 501
- Contour-parallel roughing: see "Roughing contour-parallel (G830)" on page 502
- Automatic roughing: TURN PLUS generates the work blocks for all roughing operations.
- Roughing hollowing
  - Residual longitudinal roughing: see "Residual roughing – longitudinal" on page 503
  - Residual transverse roughing: see "Residual roughing – transverse" on page 504
  - Residual contour-parallel roughing: see "Residual roughing – contour-parallel" on page 505
  - Automatic hollowing: TURN PLUS first selects a tool for rough-machining and then a tool for removing the residual material in the opposite machining direction.
- Roughing hollowing (neutral tool): see "Roughing hollowing – neutral tool (G835)" on page 506

Hollowing – Fundamentals

Using the "Roughing hollowing" (residual roughing) function, you can remove residual material from descending contours.

If no cutting limits are defined, TURN PLUS machines in the selected area. To avoid collision, the selected machining area is reduced by the **cutting limitation** function. The machining cycle accounts for the safety clearance (SAR, SIR – machining parameter 2) in front of the remaining material.

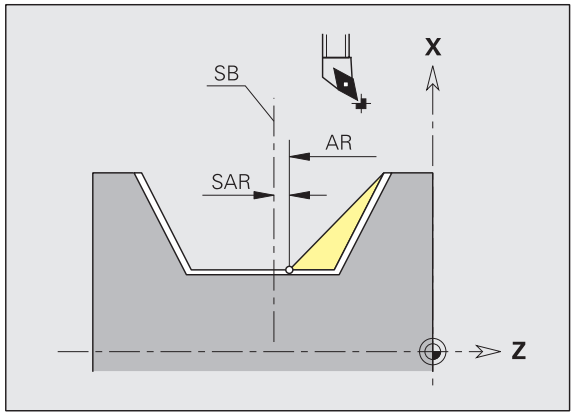
Danger of collision

The residual material is machined without monitoring for collision. Check the cutting limits and the "Angle of approach" cycle parameter.

The "Automatic hollowing" function can only be used for machining recesses. Relief turns are machined using the standard roughing cycle. TURN PLUS uses the "Permissible inward copying angle EKW" (machining parameter 1) to distinguish recesses from relief turns.

Defining the cutting limitation

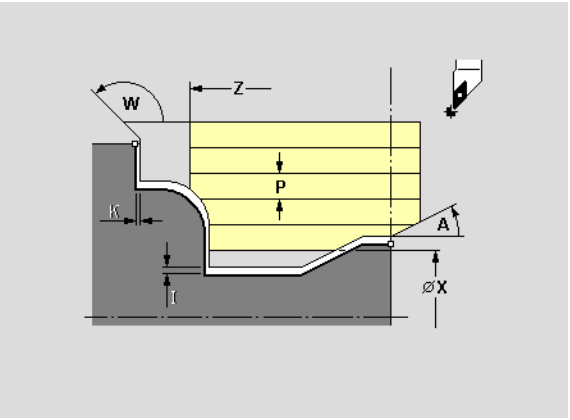
- Position the tool so that it moves on the same side as the residual material.
- Select the machining range
- Select the starting point of the remaining material as the position of the cutting limitation.



AR	Starting point of residual material
SAR	External safety clearance
SB	Cutting limit

Roughing longitudinal (G810)

The IWG generates the cycle G810 for the selected contour area.



Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 0°/180°)

W Departure angle – reference: Z axis (default 90°/270°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

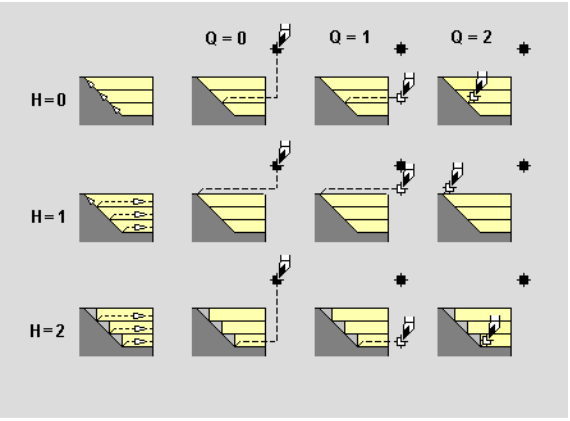
H Type of departure (type of contour smoothing)

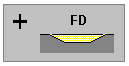
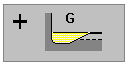
- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45° – no contour smoothing

Q Type of retraction at end of cycle

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

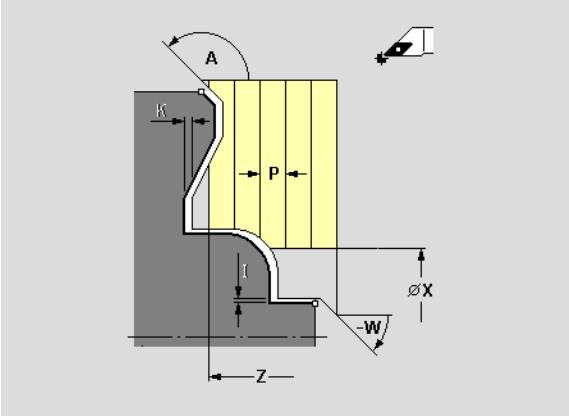
Undercut machining. The setting is made by soft key.



"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Roughing transverse (G820)

The IWG generates the cycle G820 for the selected contour area.



Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 90°/270°)

W Departure angle – reference: Z axis (default 0°/180°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

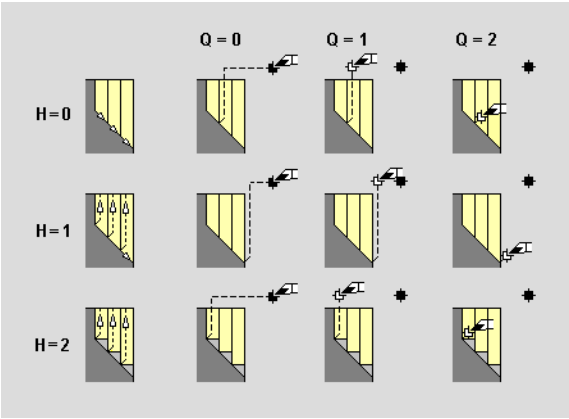
H Type of departure (type of contour smoothing)

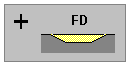
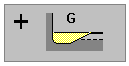
- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45° – no contour smoothing

Q Type of retraction at end of cycle

- Q=0: Return to starting point (first X, then Z direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

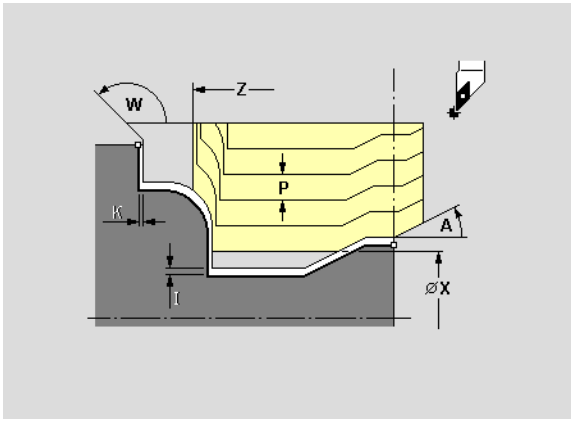
Undercut machining. The setting is made by soft key.



"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Roughing contour-parallel (G830)

The IWG generates the cycle G830 for the selected contour area.



Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 0°/180°)

W Departure angle – reference: Z axis (default 90°/270°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

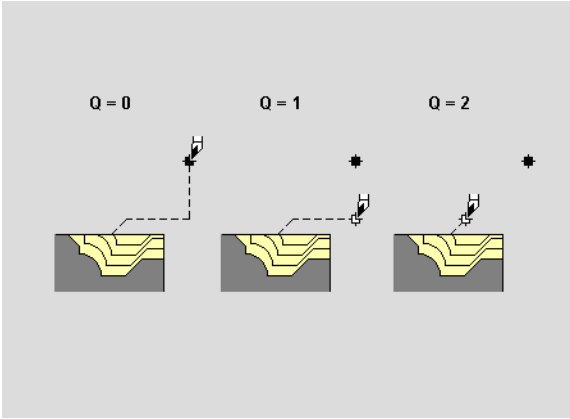
- Yes
- No

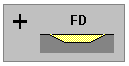
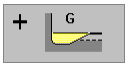
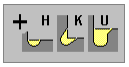
E Reduced plunging feed rate with descending contours

Q Type of retraction at end of cycle

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

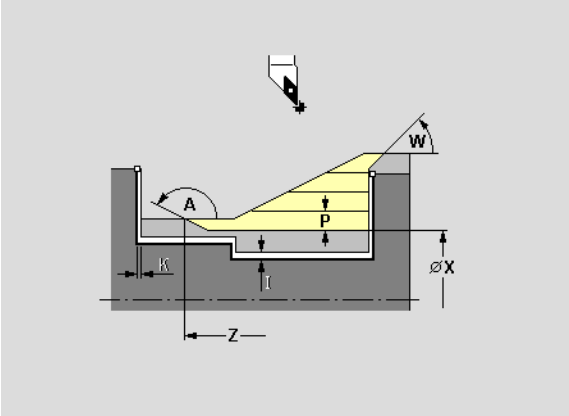
Undercut machining. The setting is made by soft key.



"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Residual roughing – longitudinal

The IWG generates the cycle G810 for removing residual material.



Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 0°/180°)

W Departure angle – reference: Z axis (default 90°/270°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

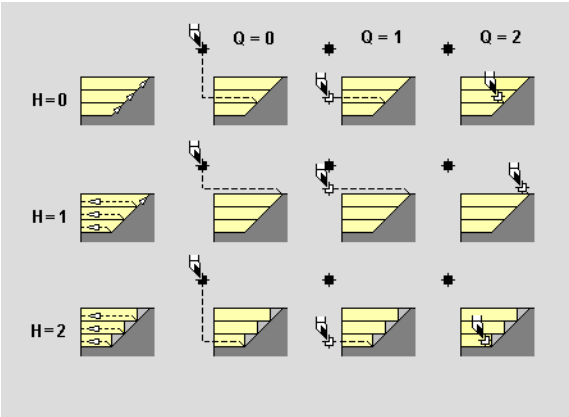
H Type of departure (type of contour smoothing)

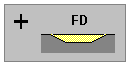
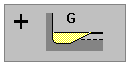
- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45° – no contour smoothing

Q Type of retraction at end of cycle

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

Undercut machining. The setting is made by soft key.



"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Residual roughing – transverse

The IWG generates the cycle G820 for removing residual material.

Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 90°/270°)

W Departure angle – reference: Z axis (default 0°/180°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

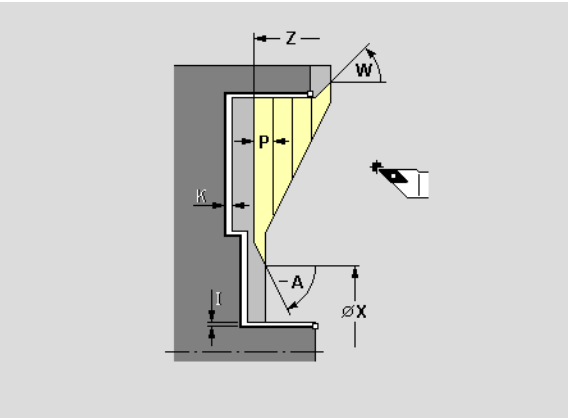
H Type of departure (type of contour smoothing)

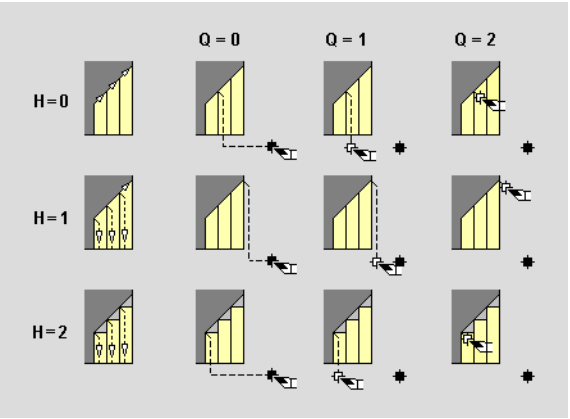
- H=0: Machines contour outline after each pass
- H=1: Retracts at 45°; contour smoothing after last pass
- H=2: Retracts at 45° – no contour smoothing

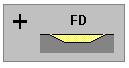
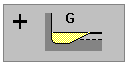
Q Type of retraction at end of cycle

- Q=0: Return to starting point (first X, then Z direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

Undercut machining. The setting is made by soft key.





"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Residual roughing – contour-parallel

The IWG generates the cycle G830 for removing residual material.

Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 0°/180°)

W Departure angle – reference: Z axis (default 90°/270°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

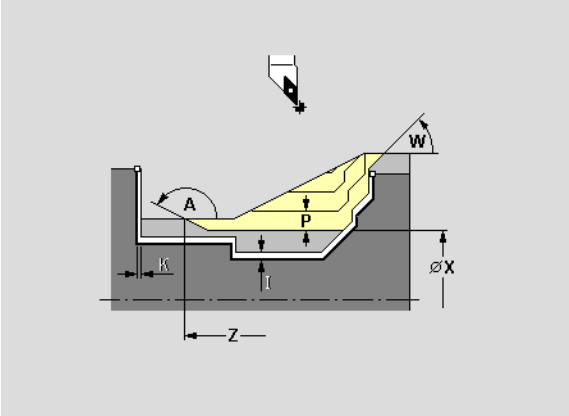
- Yes
- No

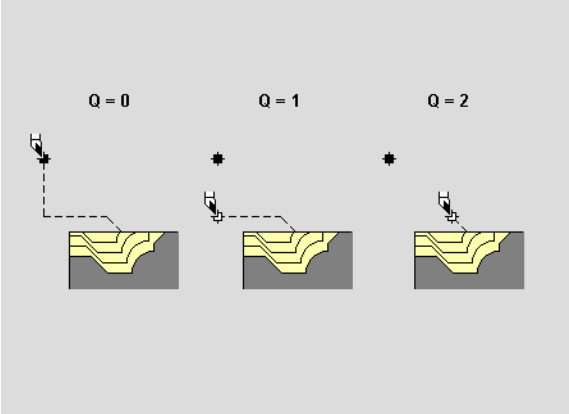
E Reduced plunging feed rate with descending contours

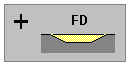
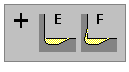
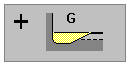
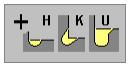
Q Type of retraction at end of cycle

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

Undercut machining. The setting is made by soft key.

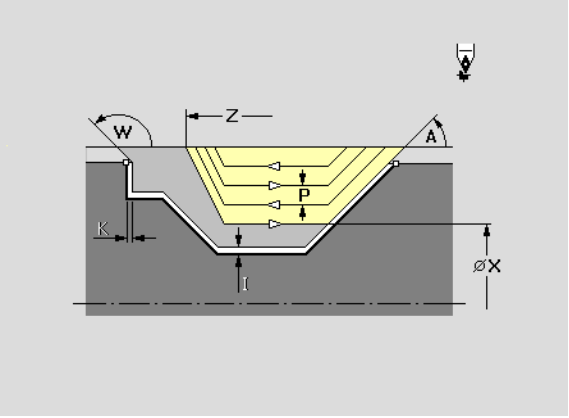




"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Roughing hollowing – neutral tool (G835)

The IWG generates the cycle G835 for the selected contour area.



Parameters

P Cutting depth (maximum infeed)

A Approach angle – reference: Z axis (default 0°/180°)

W Departure angle – reference: Z axis (default 90°/270°)

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

K Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

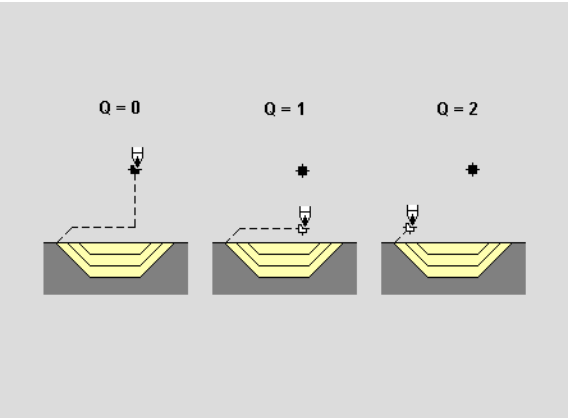
Bidirectional machining

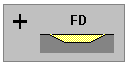
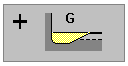
- Yes: Machining using cycle G835
- No: Machining using cycle G830

Q Type of retraction at end of cycle

- Q=0: Returns to starting point (first Z, then X direction)
- Q=1: Positions in front of the finished contour
- Q=2: Retracts to safety clearance and stops

Undercut machining. The setting is made by soft key.



"Roughing" soft keys	
	Longitudinal/constant oversize
	FD relief turn machining
	E and F undercut machining
	G undercut machining
	H, K and U undercut machining

Overview of recessing operations

The IWG presents the following recessing operations for selection ("Recessing" drop-down menu):

- Contour recessing (see "Contour recessing radial/axial (G860)" on page 508)
  - Radial contour recessing
  - Axial contour recessing
  - Automatic contour recessing
- Recessing (see "Recessing radial/axial (G866)" on page 509)
  - Radial recessing
  - Axial recessing
  - Automatic recessing
- Recess turning (see "Recess turning radial/axial (G869)" on page 510)
  - Radial recess turning
  - Axial recess turning
  - Automatic recess turning
- Parting (see "Parting" on page 512)
- Prepare parting/rear-face machining (see "Parting and workpiece transfer" on page 513)

Automatic recessing: TURN PLUS generates the work blocks for all radial and axial recessing operations.

Contour recessing radial/axial (G860)

The IWG generates the cycle G860 for the form elements "Recess general" and "Relief turn" (recess type F) and for freely defined recesses.

Parameters

X Cutting limit

Z Cutting limit

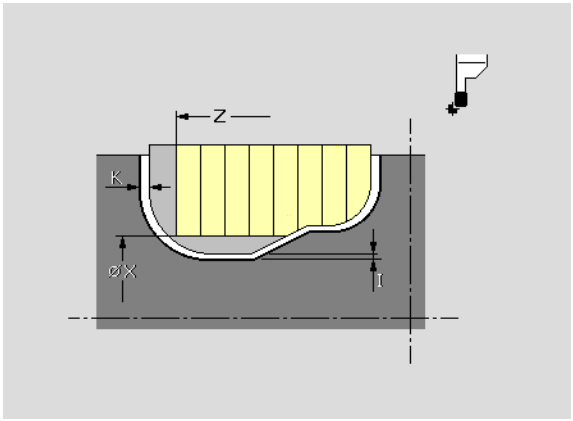
I Depending on the soft-key setting:

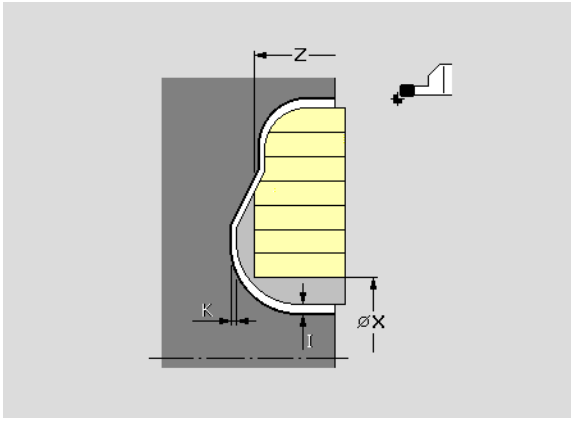
- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

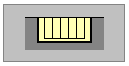
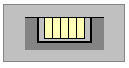
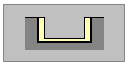
K Transverse oversize

Cycle sequence (define by soft key)

- Precutting and finishing in one machining cycle
- Only precutting
- Only finishing





"Recessing sequence type" soft keys	
	Define longitudinal/constant oversize
	Precutting and finishing
	Precutting
	Finishing

Recessing radial/axial (G866)

The IWG generates the cycle G866 for the form elements "Recess type D" (sealing ring) and "Recess type S" (guarding ring).

If an oversize has been specified, TURN PLUS first rough-machines and then finish-machines the recess.

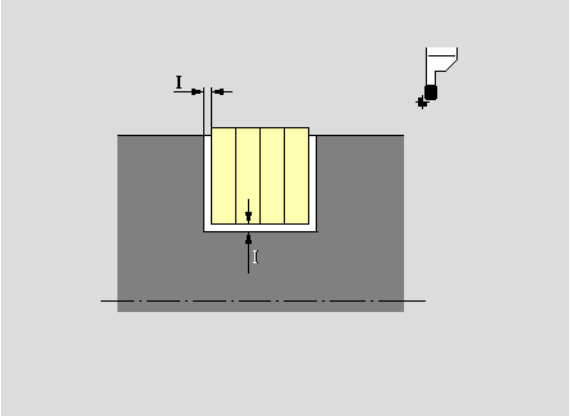
The dwell time is taken into account:

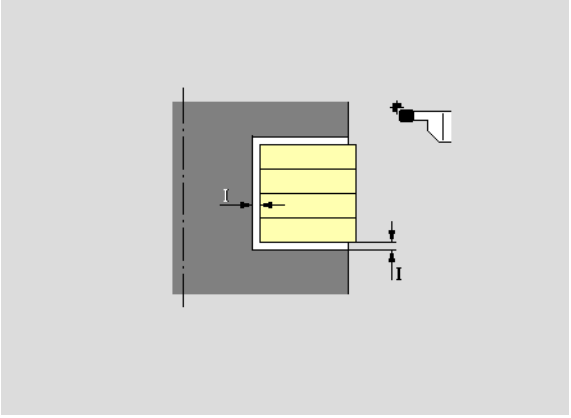
- Only for finishing, if the oversize is defined
- For each recess, if the oversize is not defined

Parameters

I Oversize (longitudinal and transverse)

E Dwell time

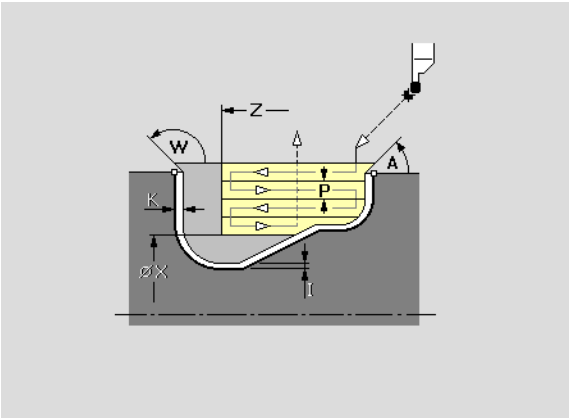


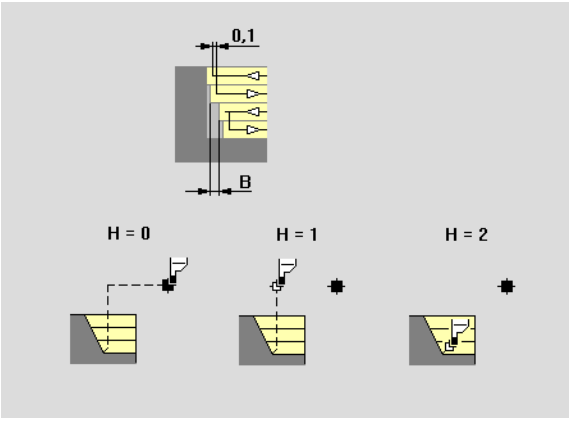


Recess turning radial/axial (G869)

The IWG generates the cycle G869 for the selected contour area (machining with alternating recessing and roughing passes).

The parameters for radial and axial recess turning are identical, except for the reference axis of the approach and departure angles. For axial recess turning: see "Recess turning axial (G869)" on page 511





Parameters

P Maximum cutting depth

R Depth compensation

Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct this infeed error with the "turning depth compensation factor." The compensation is usually determined empirically.

B Offset width

After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the "offset width." Each time the system switches from turning to recessing on this side, the path is reduced by "B," in addition to the previous offset. After precutting, the remaining material is removed with a single cut.

A Approach angle (default: opposite to recessing direction)

- Radial: Reference axis Z
- Axial: Reference axis X

W Departure angle (default: opposite to recessing direction)

- Radial: Reference axis Z
- Axial: Reference axis X

X Cutting limit

Z Cutting limit

I Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

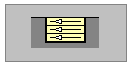
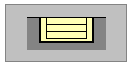
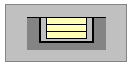
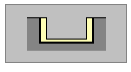
K Transverse oversize

S (Unidirectional/)Bidirectional precutting (setting by soft key)

- Yes (S=0): Bidirectional
- No (S=1): Unidirectional in the direction specified by the machining range selection

O Recessing feed rate (default: active feed rate)

E Finishing feed rate (default: active feed rate)

"Recess turning" soft keys	
	Longitudinal/constant oversize
	Unidirectional/Bidirectional
	Precutting and finishing
	Precutting
	Finishing

Parameters

H Type of retraction at end of cycle

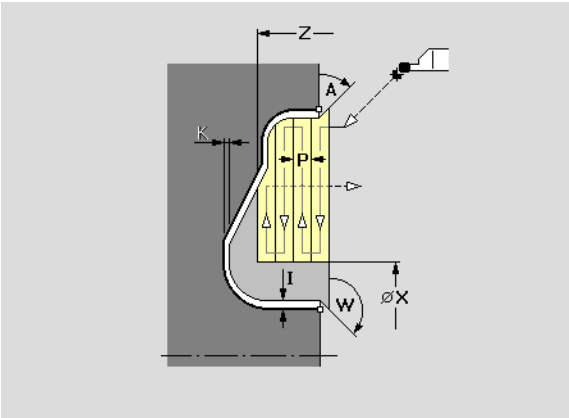
- H=0: Returns to starting point (first X, then Z direction)
- H=1: Positions in front of the finished contour
- H=2: Retracts to safety clearance and stops

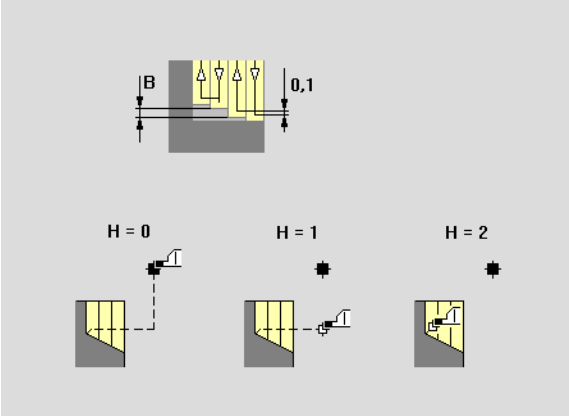
Process (setting by soft key):

- Precutting and finishing in one machining cycle
- Only precutting
- Only finishing

Recess turning axial (G869)

When defining a "Recess turning axial" cycle, pay attention to the correct reference axis for the approach and departure angles. The other parameters are the same as for "Recess turning radial" (see "Recess turning radial/axial (G869)" on page 510).

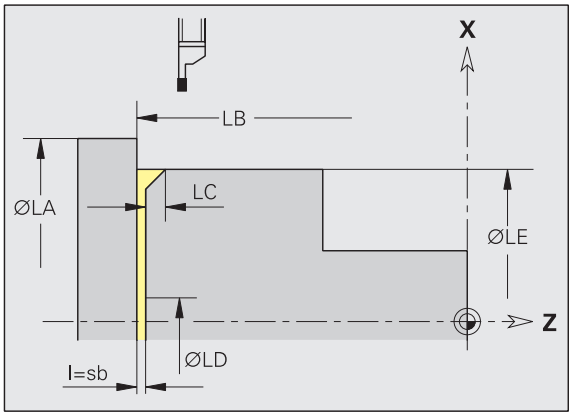




Parting

For parting, the AWG activates the **expert program** entered in machining parameter 21 – "SP 100098."

TURN PLUS determines the parameters as far as possible and enters them as default values. Check, edit or enter the values.



Parameters

LA Bar diameter

LB Starting point in Z. TURN PLUS uses the position that you defined when selecting the machining range.

LC Chamfer/Rounding

- < 0: Chamfer width
- > 0: Rounding radius

LD Feed rate reduction from position X. The reduced feed rate is defined in the expert program.

LE Diameter of finished part for determining the position of the chamfer/rounding

LF Inside diameter. The expert program traverses beyond this position to ensure safe parting.

- = 0: for a solid workpiece
- > 0: for a pipe/tube

LH Safety clearance to starting position X

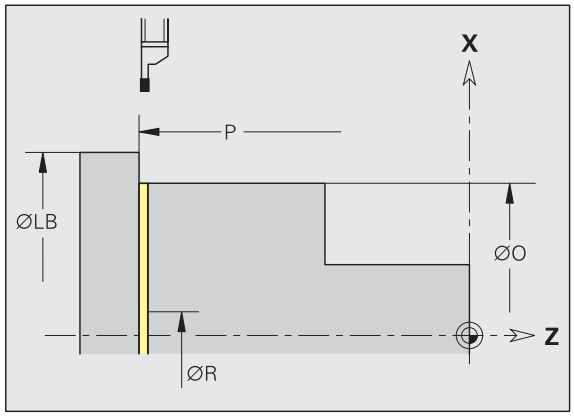
l Width of parting tool. This parameter is usually not evaluated.

- **Select the machining range:** Select the vertical element, at which part is to be parted and the chamfer/rounding is to be machined.
- Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

Parting and workpiece transfer

For parting with workpiece transfer, TURN PLUS activates an **expert program** from machining parameter 21. Which expert program is activated depends on the spindle entries specified for "1st setup .." and "2nd setup .." in the program head:

- Same spindle (manual rechucking): Entry from "SP-ABHAND."
- Different spindles (workpiece transfer to opposing spindle): Entry from "SP-UMKOMPLA."



Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

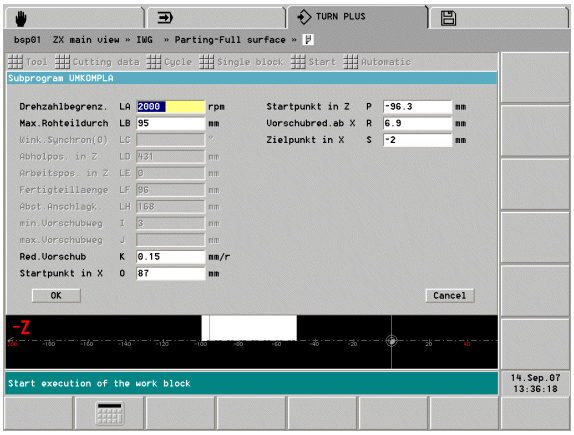
Process of parting and workpiece transfer:

- Select the vertical element at which the workpiece is to be parted. TURN PLUS opens the dialog box of the expert program.
- Check/edit the parameters.
- TURN PLUS executes the parting operation.
- Define the chucking data and position for the second setup.
- Check/edit the "workpiece transfer" parameters.
- TURN PLUS transfers the workpiece.

Expert program “UMKOMPLA”

The expert program entered in SP-UMKOMPLA (machining parameter 21) parts the workpiece and transfers it to the opposing spindle.

TURN PLUS entered the calculated parameters as proposed values. Check, edit or enter the values.



Parameters (example)

LA Spindle speed limitation for the parting process

LB Maximum diameter of workpiece blank (proposed value: from the tool description)

K Reduced feed rate for the parting process

- 0: No feed rate reduction
- >0: (Reduced) feed rate

O Start point in X for the parting process (proposed value: from the tool description)

P Start point in Z for the parting process (proposed value: vertical element from the “selection”)

R Feed rate reduction in X. From this position onwards, the tool is traversed at the reduced feed rate.

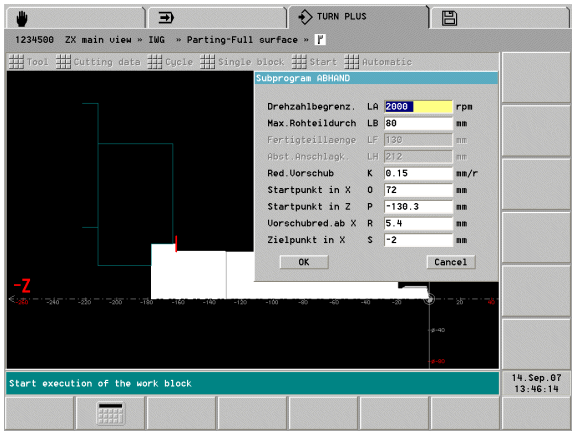
S Target position in X. End point of the parting process.

Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

Expert program “ABHAND”

The expert program entered in SP-ABHAND (machining parameter 21) parts the workpiece and supports the manual rechucking of the workpiece for machining the rear face on machines with one spindle.

TURN PLUS entered the calculated parameters as proposed values. Check, edit or enter the values.



Parameters (example)

LA Spindle speed limitation for the parting process

LB Maximum diameter of workpiece blank

K Reduced feed rate for the parting process

- 0: No feed rate reduction
- >0: (Reduced) feed rate

O Start point in X for the parting process (proposed value: from the tool description)

P Start point in Z for the parting process (proposed value: vertical element from the “selection”)

R Feed rate reduction in X. From this position onwards, the tool is traversed at the reduced feed rate.

S Target position in X. End point of the parting process.

Expert programs are provided by the machine tool builder. Refer to the machine manual for the meanings of the parameters and the sequence of the program.

Overview of drilling operations

The IWG presents the following drilling operations for selection ("Drilling" drop-down menu):

- Centric predrilling: see "Centric predrilling (G74)" on page 517
- Centering
- Drilling
- Countersinking
- Counterboring
- Reaming: see "Drilling, reaming, deep-hole drilling" on page 519
- Tapping
- Special drilling
  - Special drilling > Centering and countersinking
  - Special drilling > Drilling and countersinking
  - Drilling with thread
  - Drilling with reaming
- Automatic drilling: Accounts for the following form elements: Bore holes, single holes and hole patterns.

For

- Stationary tools: Drilling at turning center
- Driven tools: C-axis machining

Centric predrilling – Automatic: This cycle performs the complete predrilling operation, including tool changes that might be necessary because of different diameters.

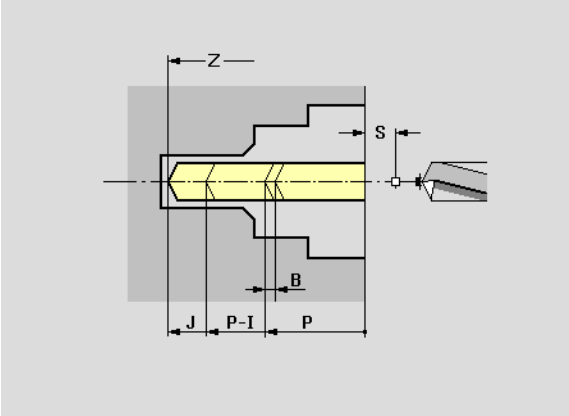
With the following drilling operations, the IWG generates

- Cycle G72 (see "Centering, countersinking (G72)" on page 518):
  - Centering
  - Countersinking
  - Counterboring
  - Special drilling > Centering and countersinking
  - Special drilling > Drilling and countersinking
- Cycle G73 (see "Tapping" on page 520):
  - Tapping
  - Drilling with thread
- Cycle G71 or G74 (see "Drilling, reaming, deep-hole drilling" on page 519):
  - Drilling
  - Drilling with reaming

Centric predrilling (G74)

The IWG generates the cycle G74 for the selected contour area (predrilling at turning center using stationary tools).

Select the machining range: Select all contour elements encompassing the bore hole. If required, the bore hole can be limited with "drilling limitation Z."



Parameters

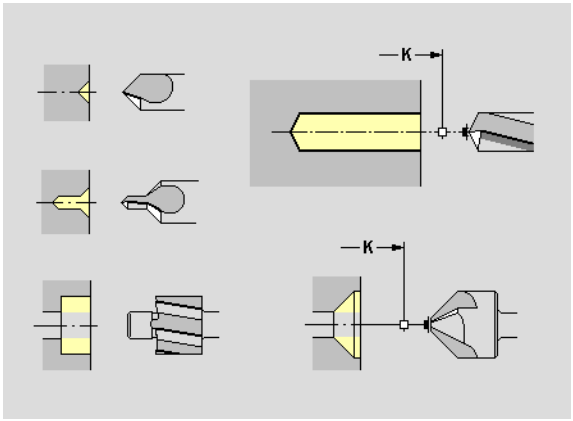
Z	Drilling limitation
S	Safety clearance (generates "Safety clearance G47" before the cycle)
P	1st hole depth
J	Minimum hole depth
I	Reduction value
B	Retraction distance (default: Retract to starting point of hole)
E	Dwell time (for chip breaking at end of hole)

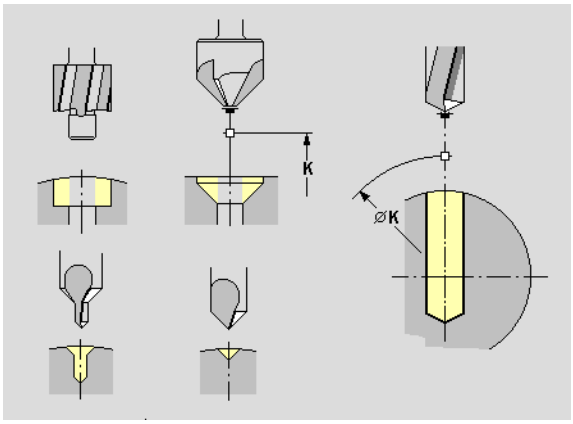
Position the drill with "Cycle > Approach" to the turning center.

Centering, countersinking (G72)

The IWG generates the cycle G72 for the following drilling operations:

- Centering
- Countersinking
- Counterboring
- Centering and countersinking (special drilling)





Parameters

K Retraction plane (default: Return to the starting position or to the safety clearance)

D Retract ("Continue" soft key)

- At the programmed feed rate
- At rapid traverse

E (Dwell time for) cutting free

Drilling, reaming, deep-hole drilling

The IWG generates the cycle G71 for the following drilling operations:

- Drilling
- Reaming
- Drilling with reaming (special drilling)

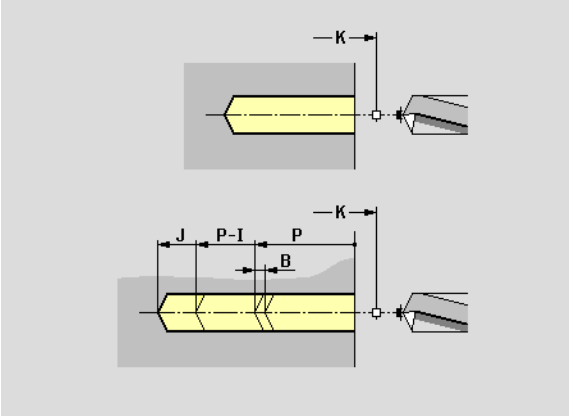
Parameters

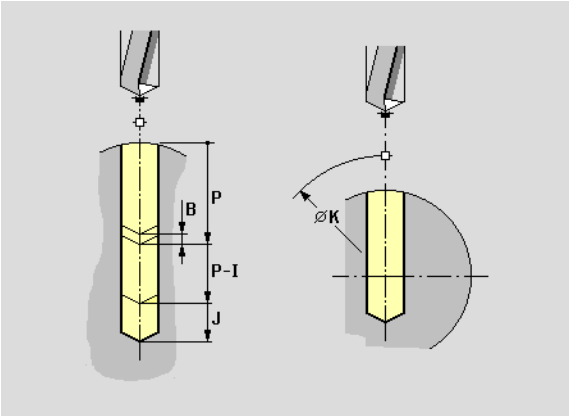
K	Retraction plane (default: Return to the starting position or to the safety clearance)
D	Retract (“Continue” soft key) ■ At the programmed feed rate ■ At rapid traverse
E	(Dwell time for) cutting free
F50%	Feed rate reduction – see soft key table
P	1st hole depth
J	Minimum hole depth
I	Reduction of depth (reduction value)
B	Retraction value (return distance) (default: Retract to starting point of hole)

If you enter the parameters for deep-hole drilling, the IWG generates the cycle G74.

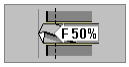
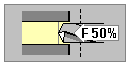
Feed rate reduction: For blind drilling and/or through drilling you can define a feed rate reduction of 50%. The feed rate reduction for through drilling is switched on depending on the type of tool:

- Boring bar with indexable inserts and twist drills with 180° drill angle: Feed-rate reduction switched on at drill tip – 2 * safety clearance
- Other drills: Drill end – length of first cut – safety clearance (length of first cut=drill tip; safety clearance: see “Drilling – Safety clearances” on page 601 or G47, G147).





“Feed rate reduction” soft keys

	Feed rate reduction for through drilling
	Feed rate reduction for blind drilling
	Feed rate reduction for blind drilling with drill angle < 180°

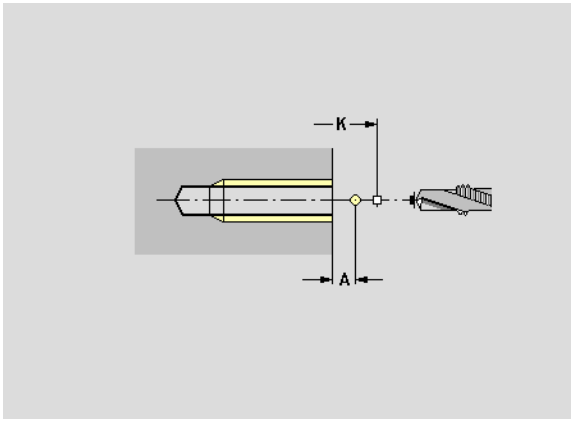
Tapping

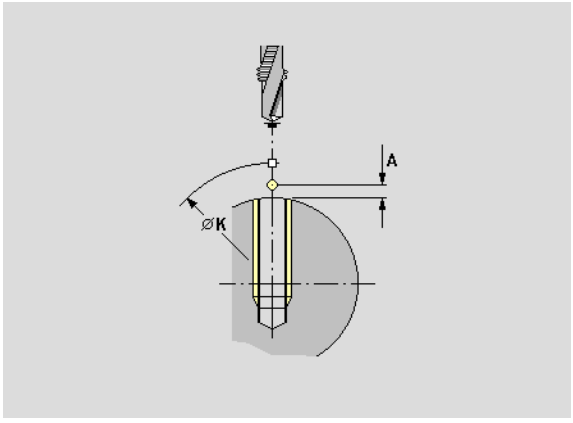
The IWG generates the cycle G73 for the following drilling operations:

- Tapping
- Drilling with thread (special drilling)

Parameters

K	Retraction plane (default: Return to the starting position or to the safety clearance)
D	Retract (“Continue” soft key) ■ At the programmed feed rate ■ At rapid traverse
A	Slope length (default: Machining parameter 7 “Thread starting length [GAL]”)
S	Retraction speed (default: Tapping speed)





Finishing

The IWG presents the following finishing operations for selection ("Finishing" drop-down menu):

- Finishing using cycle G890:
  - Contour machining
  - Residual-contour machining
  - Finishing hollowing (neutral tool)
- Finishing using special functions:
  - Clearance turning: see "Finishing – Clearance turning" on page 524
  - Undercuts: see "Finishing – undercut" on page 524

Parameters

X Cutting limit

Z Cutting limit

L Depending on the soft-key setting:

- Longitudinal oversize
- Constant oversize (generates "Oversize G58" before the cycle)

P Transverse oversize

Plunge-cutting (machine descending contours)?

- Yes
- No

E Reduced plunging feed rate with descending contours

Approach

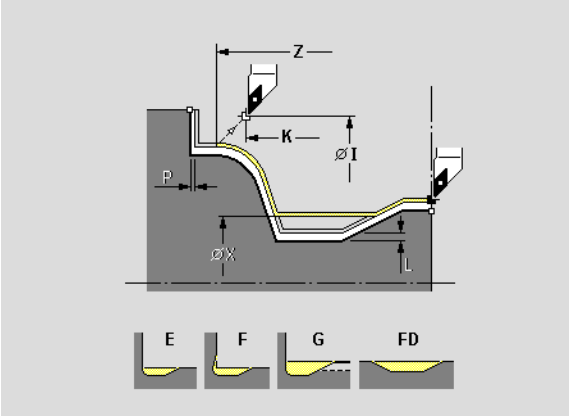
- Yes: "Mode of approach Q" (define by soft key)
- No (Q=3): Tool is located near the starting point of the contour area

Q Type of approach (see soft key table)

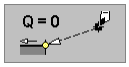
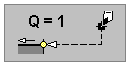
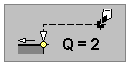
- Q=0: The IWG checks the following types of approach:
  - Diagonal approach
  - First X, then Z direction
  - Equidistant around the barrier
  - Omission of the first contour element if the starting position is inaccessible
- Q=1: First X, then Z direction
- Q=2: First Z, then X direction

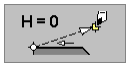
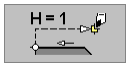
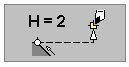
Retract

- Yes: "Type of retraction H" (define by soft key)
- No (H=4): Tool remains on the end coordinate



Soft keys	
	Longitudinal/constant oversize

"Approach" soft keys	
	Automatic selection of the mode of approach
	First X, then Z direction
	First Z, then X direction

"Retract" soft keys	
	Diagonally to retraction position
	First X, then Z direction
	First Z, then X direction
	Retract to safety clearance

Parameters

H Type of retraction. The tool retracts at 45° in the opposite direction to the machining direction. The subsequent retraction path is defined by H:

- H=0: Diagonally to retraction position
- H=1: First X, then Z direction
- H=2: First Z, then X direction
- H=3: Backs off at feed rate to the safety clearance

I Retraction position with H=0, 1, 2

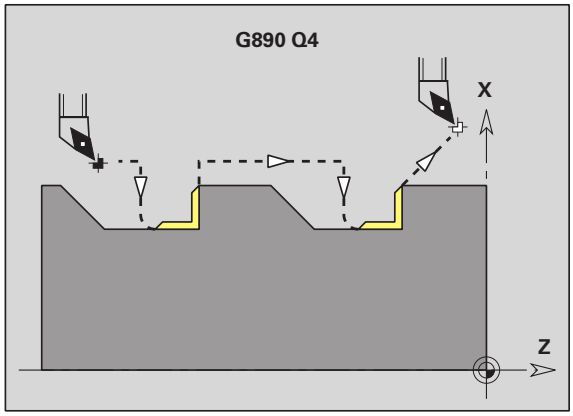
K Retraction position with H=0, 1, 2

Form element machining: Setting by soft key

The display field shows the form elements to be machined (for abbreviations see soft key table). The following form elements are always machined:

- C: Chamfer
- R: Rounding
- PT: Fit
- GW: Thread

Residual-contour machining: Using the "Residual-contour machining" function, you can remove residual material from descending contours (see illustration of "G890 Q4"). Normally a **cutting limit** is not necessary.



Hollowing: The IWG machines recess areas determined with the aid of the "inward copying angle" (recesses: inward copying angle <= mtw). For Finishing – Hollowing, you should use neutral tools.

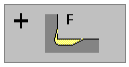
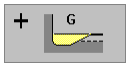
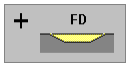
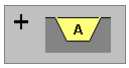
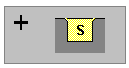
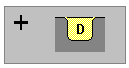
The approach strategy is predefined for remaining-contour machining and hollowing. The IWG generates the cycle G890 with "type of approach Q4."

For **chamfers/rounding arcs,** the following applies:

- The "Feed/Peak-to-valley" attribute is not programmed: The CNC PILOT automatically reduces the feed rate. At least "FMUR" revolutions (machining parameter 5) are performed.
- "Feed/Peak-to-valley" attribute programmed: No feed rate reduction.
- There is no feed-rate reduction for chamfers/rounding arcs that, due to their size, are machined with at least FMUR revolutions (machining parameter 5).

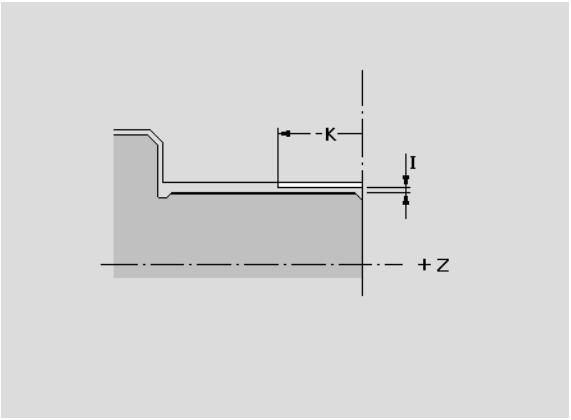
The value proposed for the retraction position I,K depends on whether you program "Cycle > Approach:"

- If programmed: Position from "Cycle > Approach"
- If not programmed: Position of the tool change point

“Form element machining” soft keys		
	Call the “Form elements” soft keys	
	Undercut type E	E
	Undercut type F	F
	Undercut type G	G
	Relief turn	FD
	Call the “Form elements” soft keys	
	Undercut type H	H
	Undercut type K	K
	Undercut type U	U
	Recess general	A
	Recess form S	S
	Recess form D	D
	Switch to previous soft-key row	

Finishing – Clearance turning

TURN PLUS executes a **measuring cut** on the selected contour element. Precondition: The contour element was assigned the "Measure" attribute (see "Machining attributes – Measure" on page 473).



Parameters

I	Oversize for measuring cut
K	Length of measuring cut
Q	Measuring loop counter (every nth workpiece is measured)

Clearance turning is performed by the **expert program** SP-MEAS01 (machining parameter 21). Parameters of the expert program: see your machine manual.

Finishing – undercut

"Finishing – undercut" can be used for machining undercuts:

- Type U
- Type H
- Type K

When machining undercut type U, adjacent transverse elements for which an oversize is taken into account are machined to finished dimensions.

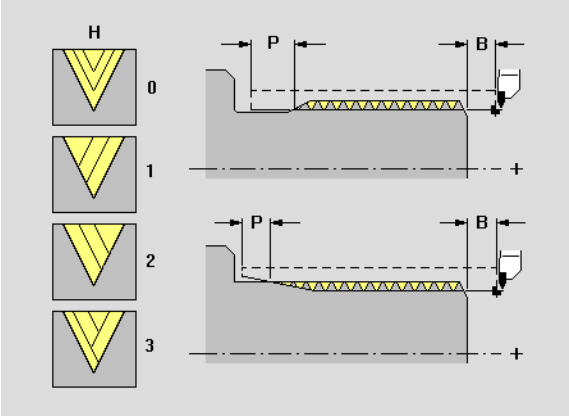
Operation:

- Select the tool
- Select the machining range
- Press "Start"

You cannot influence an undercut operation (the "Cycle > Cycle parameter" menu item cannot be selected).

Thread machining (G31)

The IWG generates the cycle G31 for the selected thread.



Parameters

B Run-in length

- No input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses.
- No input, no undercut/recess: The CNC PILOT uses the thread starting length from machining parameter 7.

P Overrun length

- No input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses.
- No input, no undercut/recess: The CNC PILOT uses the thread run-out length from machining parameter 7.

C Starting angle, if the beginning of the thread is defined with respect to rotationally nonsymmetrical contour elements

I Maximum infeed

V Type of infeed

- V=0: Constant cross section for all cuts
- V=1: Constant infeed
- V=2 (distribution of remaining cuts): If the division thread depth/infeed provides a remainder, the first infeed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
- V=3 (EPL method): Infeed is calculated from the pitch and spindle speed

H Type of offset used for the individual infeeds for smoothing the thread sides

- H=0: Without offset
- H=1: Offset from left
- H=2: Offset from right
- H=3: Tool is offset alternately from the right and left

Q Number of air cuts after the last cut (for reducing the cutting pressure in the thread base)

Danger of collision!

An excessive overrun length P might cause a collision. You can check the overrun length during the simulation.

Overview of milling operations

The IWG presents the following milling operations for selection ("Milling" drop-down menu):

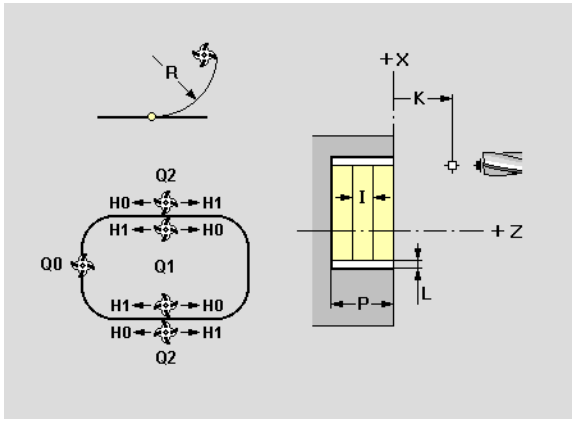
- Contour milling (see "Contour milling – Roughing/Finishing (G840)" on page 527)
  - Roughing
  - Finishing
- Area milling (see "Pocket milling – Roughing/Finishing (G845/ G846)" on page 531)
  - Roughing
  - Finishing
- Deburring: see "Deburring (G840)" on page 529
- Engraving: see "Engraving (G840)" on page 530
- Automatic milling
  - Roughing
  - Finishing

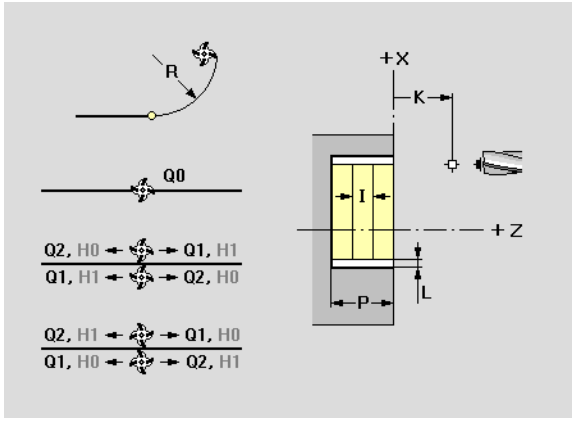
The IWG machines the milling contours of the following reference planes:

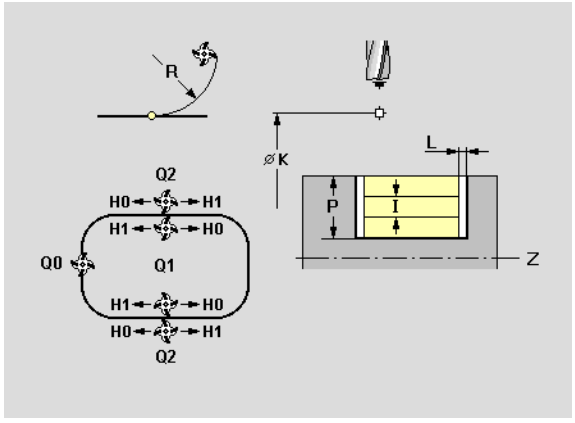
- STIRN [FRONT]
- RUECKSEITE [REAR SIDE]
- MANTEL [SURFACE]

Contour milling – Roughing/Finishing (G840)

The IWG generates the cycle G840, including the following parameters, for the selected open or closed contour.







Parameters

K Retraction plane (default: back to starting position)

- Front/read face: Position in Z direction
- Lateral surface: Position in X direction (diameter)

Q Cutter position

- Q=0: Milling center on the contour
- For closed contours:
  - Q=1: Inside milling
  - Q=2: Outside milling
- For open contours:
  - Q=1: Left of the contour (reference: machining direction)
  - Q=2: Right of the contour (reference: machining direction)

H Cutting direction

- H=0: Up-cut milling
- H=1: Climb milling

R Approach radius

- R=0: Approach contour element directly
- R>0: Approaching/departing radius that connects tangentially to the contour element
- R<0 for inside corners: Approaching/departing radius that connects tangentially to the contour element
- R<0 for outside corners: Contour element is approached/departed tangentially on a line

P Milling depth (overwrites the depth defined in the contour definition)

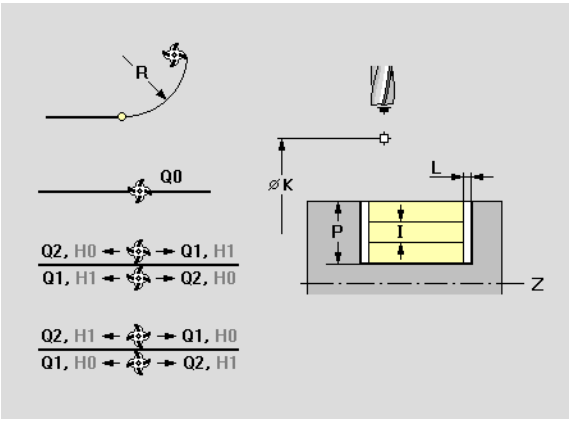
I Maximum infeed (default: milling in one infeed)

Parameters

L Oversize

The oversize “shifts” the contour depending on the cutter position Q (generates “Oversize G58” before the milling cycle):

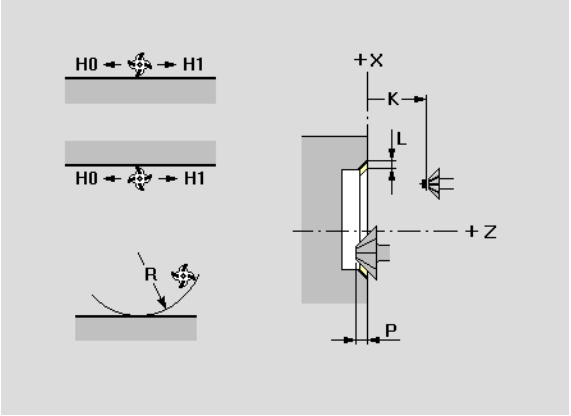
- Q=0: Oversize will be ignored
- For closed contours:
  - Q=1: Reduces the contour
  - Q=2: Enlarges the contour
- For open contours:
  - Q=1: Shift to the left
  - Q=2: Shift to the right

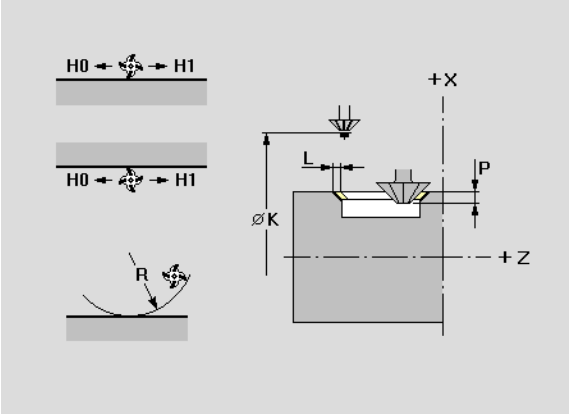


Effects of “cutter position, cutting direction and direction of tool rotation:” see “Contour milling G840—Fundamentals” on page 261.

Deburring (G840)

The IWG generates the cycle G840, including the following parameters, for the selected open or closed contour.





Parameters

K Retraction plane (default: back to starting position)

- Front/read face: Position in Z direction
- Lateral surface: Position in X direction (diameter)

Q Cutter position

- Q=0: Center of milling cutter on the contour
- For closed contours:
  - Q=1: Inside milling
  - Q=2: Outside milling
- For open contours:
  - Q=1: Left of the contour (reference: machining direction)
  - Q=2: Right of the contour (reference: machining direction)

H Mill cutting direction

- H=0: Up-cut milling
- H=1: Climb milling

R Approach radius

- R=0: Approach contour element directly
- R>0: Approaching/departing radius that connects tangentially to the contour element
- R<0 for inside corners: Approaching/departing radius that connects tangentially to the contour element
- R<0 for outside corners: Contour element is approached/departed tangentially on a line

P Milling depth – plunging depth of the tool (default: Chamfer width (from "Machining attribute – deburring") + 1 mm)

L Oversize

The oversize "shifts" the contour depending on the cutter position Q (generates "Oversize G58" before the milling cycle):

- Q=0: Oversize will be ignored
- For closed contours:
  - Q=1: Reduces the contour
  - Q=2: Enlarges the contour
- For open contours:
  - Q=1: Shift to the left
  - Q=2: Shift to the right



Deburring: The **chamfer width** is defined as machining attribute.

Engraving (G840)

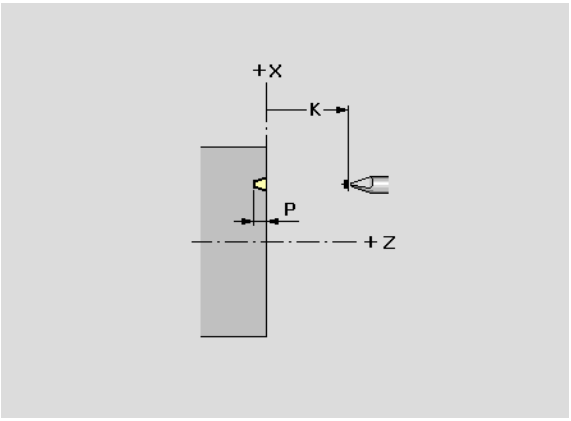
The IWG generates the cycle G840, including the following parameters, for the selected open or closed contour.

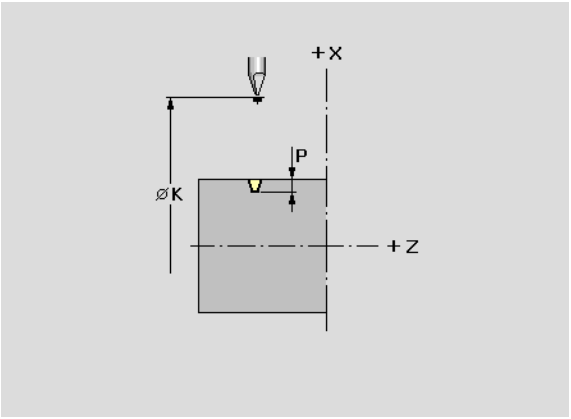
Parameters

K Retraction plane (default: back to starting position)

- Front/read face: Position in Z direction
- Lateral surface: Position in X direction (diameter)

P Milling depth – plunging depth of the tool





Pocket milling – Roughing/Finishing (G845/ G846)

The IWG generates one of the following cycles for the selected (closed) milling contour:

- Pocket milling > Roughing: G845
- Pocket milling > Roughing: G846

Parameters

J Retraction plane (default: back to starting position)
- Front/read face: Position in Z direction
- Lateral surface: Position in X direction (diameter)

Q Machining direction
- From the inside toward the outside (Q=0)
- From the outside toward the inside(Q=1)

H Cutting direction
- H=0: Up-cut milling
- H=1: Climb milling

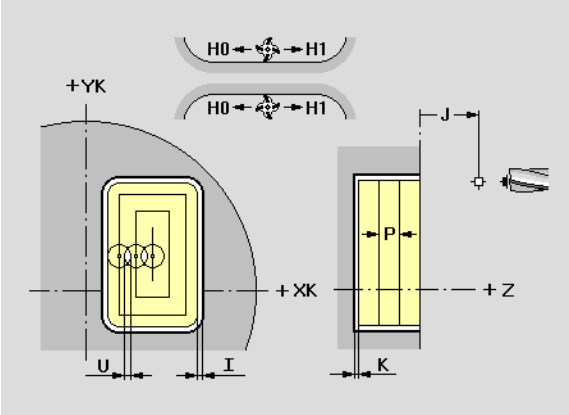
U Overlap factor

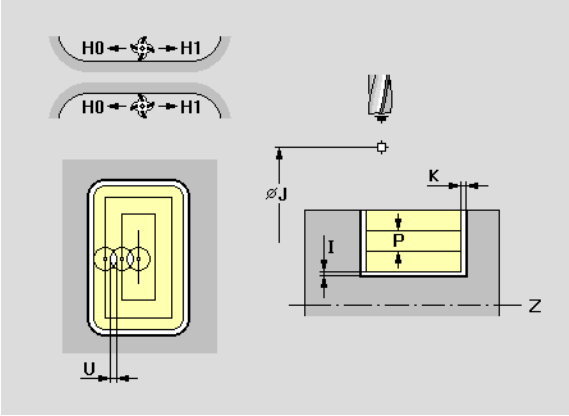
Range: 0 <= U <= 0.9; (0=no overlap)

P Maximum infeed in the milling plane

I Oversize in X direction (omitted for finishing)

K Oversize in Z direction (omitted for finishing)





Special machining (SM)

"Special machining" defines a work block that is integrated in the working plan. In special machining you add paths of traverse, subprogram calls or G/M functions (example: use of tool handling systems).

Defining tool paths at feed rate or rapid traverse

Select "Special machining > Free input" in the IWG menu.

Press "Tool."

Select and position the tool.

Select "Single block > Rapid traverse G0." TURN PLUS opens the "Rapid traverse G0" dialog box.

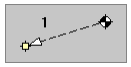
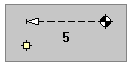
Define the target position and strategy for the path of traverse (see soft key table).

Select "Cutting data." Check/optimize the cutting data suggested by TURN PLUS.

Select "Single block > Linear path G1." TURN PLUS opens the "Linear path G1" dialog box.

Define the target position and strategy for the path of traverse (see soft key table).

If required, select "Technology > G and M functions" (or ".. > General technology") to define special functions.

Soft keys	
	Simultaneous
	First X, then Z direction
	First Z, then X direction
	Only in X direction
	Only in Z direction

Defining a subprogram call

Select “Special machining > Free input > Single block > Technology” in the IWG menu.

Select “Subprogram.” TURN PLUS opens the selection box with the available subprograms.

Select the desired subprogram and define the transfer parameters.

Select “G and M functions.”

Define the target position and strategy for the path of traverse (see soft key table).

Select “Cutting data.” Check/optimize the cutting data suggested by TURN PLUS.

Select “Single block > Linear path G1.” TURN PLUS opens the “Linear path G1” dialog box.

Define the target position and strategy for the path of traverse (see soft key table).

If required, select “Technology > G and M functions” (“..> Subprogram” or “.. > General technology“) to define special functions.

6.15 Automatic Working Plan Generation (AWG)

The **AWG** generates the work blocks of the working plan in the sequence defined in "Machining sequence." **Machining parameters** define details of machining. TURN PLUS automatically finds all the elements of a work block. An existing part machining can be continued with the AWG. Use the **machining sequence editor** to specify the machining sequence.

If the contour analysis cannot determine specific details of machining, TURN PLUS enters default values. TURN PLUS displays a warning for your information, but you cannot edit the defaults.

TURN PLUS simulates the machining in the control graphics. You can edit the sequence and representation of the control graphics in the configuration (see "Configuring the control graphics" on page 550) or set them by soft key (see "Setting the control graphics" on page 548).

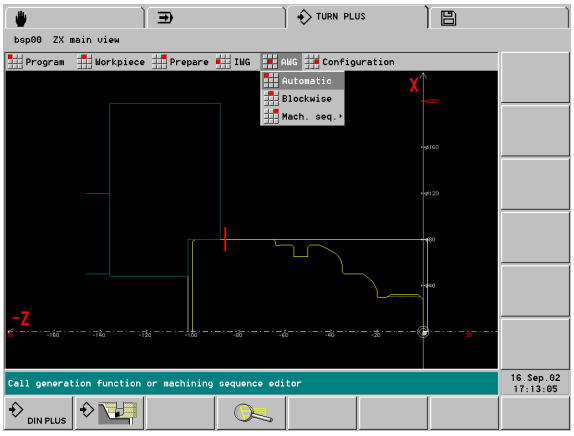
Generating a working plan

Before generating the working plan, please note:

- It is recommended that you define how to clamp the workpiece. Alternatively, TURN PLUS assumes a default type of clamping/ clamping length and adjusts the cutting limitation accordingly.
- The tool selection strategy is specified in "WD" (machining parameter 2). Define a TURN PLUS-specific turret assignment before starting the AWG.

Generating a complete working plan

Select "AWG > Automatic." TURN PLUS generates the work blocks and displays them in the control graphics.



After generation you can accept or reject the working plan.

ESC	Press the ESC key: Generation is interrupted. Work blocks that have completely been generated up to the moment of interruption are retained.

Generating a working plan block by block:

Select “AWG > Blockwise.”

TURN PLUS generates the working plan block by block and displays it in the control graphics. After generation you can accept or reject the work block.

After generation you can accept or reject the working plan.

Machining sequence – Fundamentals

TURN PLUS analyzes the contour in the sequence defined in “Machining sequence.” In this process the areas to be machined and the tool parameters are ascertained. The AWG analyzes the contour using the machining parameters.

TURN PLUS distinguishes between:

- Main machining
- Submachining
- Location (machining location)

“Submachining” and “machining location” refine the machining specification. If you do not define the submachining operation or machining location, the AWG generates machining blocks for **all** submachining operations/machining locations.

The following table lists the recommended combinations of main machining operations with submachining operations and machining locations and explains the working method of the AWG.

The following factors additionally influence the working plan:

- Geometry of the contour
- Attributes of the contour
- Tool availability
- Machining parameters

The AWG does **not** generate the work blocks if any required preparatory step is missing, or if the appropriate tool is not available, etc. TURN PLUS skips machining operations/machining sequences that do not make sense in the machining process.

You initiate **rear-face machining** with the main machining and submachining operation "Parting – Full-surface machining" or "Rechucking – Full-surface machining." You can influence rear-face machining as follows:

- Define the machining operations for the rear face after "Parting ... / Rechucking ...".
- If you define no further main machining operations after "Parting ... / Rechucking ...": TURN PLUS uses the machining sequence defined for the front face for machining the rear face.

Organizing machining sequences:

- TURN PLUS always uses the **current machining sequence**. The current machining sequence can be edited or overwritten by loading another machining sequence.
- If you load a complete program and generate a new working plan, the **current machining sequence** is taken as a basis.

Danger of collision!

When executing drilling or milling operations, TURN PLUS does not check whether the turning operation has already been completed. Ensure that turning operations precede drilling or milling operations in the machining sequence.

Editing and managing machining sequences

TURN PLUS uses the currently active machining sequence. You can adapt the machining sequences to your range of parts.

Managing the machining sequence files

Loading a machining sequence:

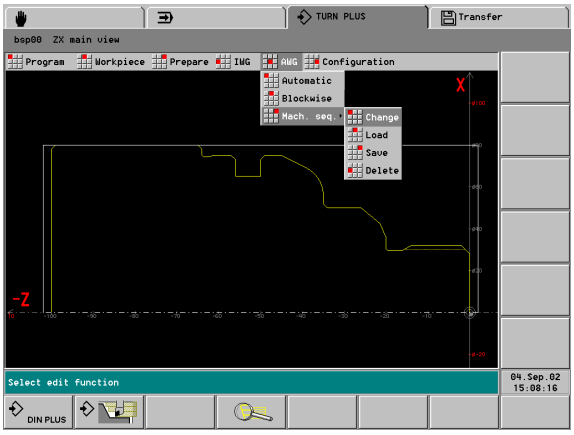
- Select "AWG > Machining sequence > Load." TURN PLUS opens the selection list with the machining sequence files.
- Select the desired file.

Saving a machining sequence:

- Select "AWG > Machining sequence > Save." TURN PLUS opens the selection list with the machining sequence files.
- Enter a new file name or overwrite an existing file.

Deleting a machining sequence:

- Select "AWG > Machining sequence > Delete." TURN PLUS opens the selection list with the machining sequence files.
- Select the file to be deleted.



Editing a machining sequence

Select “AWG > Machining sequence > Change.” TURN PLUS activates the machining sequence editor.

Select the position.

Position the cursor.

Enter a new machining operation (the new machining operation is inserted in front of the cursor).

Insert	TURN PLUS opens the “Machining sequence – editor” dialog box.

Use the arrow keys to select the main machining and submachining operation and the location. Confirm the setting with ENTER.

“OK” confirms the new machining sequence.

Editing the machining sequence

Change	TURN PLUS opens the “Machining sequence – editor” dialog box.

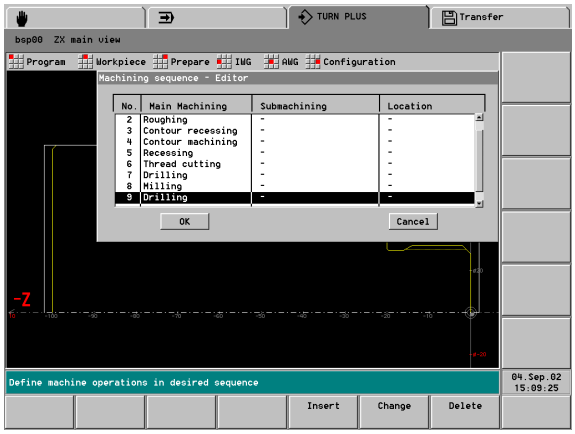
Use the arrow keys to select the main machining/submachining operation or location. Confirm the setting with ENTER.

“OK” confirms the new machining sequence.

Deleting a machining operation

Delete	Press the soft key. TURN PLUS removes the machining operation.

“OK” saves the changed machining sequence.



Overview of machining sequences

Special machining has no significance for the AWG.

Machining sequence for centric predrilling

Main machining	Submachining	Location	Description
Centric predrilling			**Contour analysis:** Determining the drilling steps
			Machining parameter: 3 – Centric predrilling
	–	–	Predrilling of 1st step
			Predrilling of 2nd step
			Finish-drilling
	Predrilling	–	Predrilling of 1st step
			Predrilling of 2nd step
	Finish-drilling	–	Finish-drilling

Machining sequence for roughing without hollowing

Main machining	Submachining	Location	Execution
Roughing (without hollowing)			**Contour analysis**: Dividing the contour into areas for longitudinal/ transverse outside machining and longitudinal/transverse inside machining according to the transverse/longitudinal ratio.
			Sequence: First outside, then inside machining
			Machining parameter: 4 – Roughing
	–	–	Transverse machining, longitudinal machining – outside and inside
	Longitudinal	–	Longitudinal machining – outside and inside
	Longitudinal	Outside	Longitudinal machining – outside
	Longitudinal	Inside	Longitudinal machining – inside
	Transverse	–	Transverse machining
	Contour-parallel	–	Contour-parallel machining – outside and inside
	Contour-parallel	Outside	Contour-parallel machining – outside
	Contour-parallel	Inside	Contour-parallel machining – inside

Machining sequence for roughing with hollowing

Main machining	Submachining	Location	Execution
(Roughing) **Hollowing**			**Contour analysis:** Using the "inward copying angle (EKW)," you can determine recess areas (undefined recesses). The machining operation is performed with one or two tools.
			Sequence: First outside, then inside machining
			Machining parameter: 1 – Global parameters for finished parts
	–	–	Longitudinal machining, transverse machining – outside and inside
	Longitudinal	Outside	Longitudinal machining – outside
	Longitudinal	Inside	Longitudinal machining – inside
	Transverse	Outside	Transverse machining – outside, front and back
	Transverse	Inside	Transverse machining – inside
	Transverse	Outside/ front	Transverse machining – outside/front
	Transverse	Outside/ back	Transverse machining – outside/back
	Neutral tool	–	Longitudinal machining, transverse machining – outside and inside
	Neutral tool	Outside	Longitudinal machining – outside
	Neutral tool	Inside	Longitudinal machining – inside
	Neutral tool	Outside/ front	Transverse machining – outside, front and back
	Neutral tool	Inside/front	Transverse machining – inside



If hollowing **precedes** recess-turning/contour-recessing in the machining sequence, recess areas are hollowed – exception: no suitable tools are available.

Machining sequence for contour machining (finishing)

Main machining	Submachining	Location	Execution
Contour machining (finishing)			**Contour analysis**: Dividing the contour into areas for outside and inside machining.
			Sequence: First outside, then inside machining
			Machining parameter: 5 – Finishing
	Contour-parallel	–	Outside/inside machining
	Contour-parallel	Outside	Outside machining
	Contour-parallel	Inside	Inside machining
	Neutral tool	–	Outside/inside machining
	Neutral tool	Outside	Outside machining
	Neutral tool	Inside	Inside machining
	Neutral tool	Outside/ front	Machining the front/back – outside
	Neutral tool	Inside/front	Machining the front – inside



- **Undefined recesses** are machined if previously rough-machined.
  - Contour-parallel submachining (standard tools): Finish-machining according to the "Hollowing" method.
  - Submachining with neutral tool: Finish-machining using one tool.
- **Clearance machining:** The AWG takes contour elements with the "Measure" machining attribute into account in the finish-machining operation.

Machining sequence for recess turning

Main machining	Submachining	Location	Execution
Recess turning			**Contour analysis**: ■ Without previous **roughing operation**: The complete contour, including recess areas (undefined recesses), is machined. ■ With previous **roughing**: Recess areas (undefined recesses) are determined and machined according to the "inward copying angle (EKW)." **Sequence**: First outside, then inside machining **Machining parameter:** 1 Global parameters for finished parts
	–	–	Radial/axial machining – outside and inside
	Contour-parallel	Outside	Radial machining – outside
	Contour-parallel	Inside	Radial machining – inside
	Contour-parallel	Outside/ front	Axial machining – outside
	Contour-parallel	Inside/front	Axial machining – inside



- If recess turning **precedes** hollowing in the machining sequence, recess areas are machined by recess turning – exception: no suitable tools are available.
- Recess turning – contour turning are used alternatively.

Machining sequence for contour recessing

Main machining	Submachining	Location	Execution
Contour recessing			**Contour analysis:** Recess areas (recesses) are determined and machined according to the "inward copying angle (EKW)."
			Sequence: First outside, then inside machining
			Machining parameter: 1 Global parameters for finished parts
	–	–	Radial/axial machining – outside and inside Shaft machining: Axial machining on the outside is performed on front and back
	Contour-parallel	Outside	Radial machining – outside Shaft machining: Front and back
	Contour-parallel	Inside	Radial machining – inside
	Contour-parallel	Outside/ front	Axial machining – outside
	Contour-parallel	Inside/front	Axial machining – inside



- If contour recessing **precedes** hollowing in the machining sequence, recess areas are machined by contour recessing – exception: no suitable tools are available.
- Recess turning – contour turning are used alternatively.

Machining sequence for recessing

Main machining	Submachining	Location	Execution
Recessing			**Contour analysis:** Determining the "Recess" form elements: ■ Type S (guarding ring – recess type S) ■ Type D (sealing ring – recess type D) ■ Type A (recess general) ■ Type FD (relief turn F) – FD is only machined using "Recessing" if the "inward copying angle (EKW) <= mtw."
			Sequence: First outside, then inside machining
			Machining parameter (with type FD): 1 Global parameters for finished parts
	–	–	All recess types; radial/axial machining; outside and inside
	Type S, D, A, FD (*)	Outside	Radial machining – outside
	Type S, D, A, FD (*)	Inside	Radial machining – inside
	Type A, FD (*)	Outside/ front	Axial machining – outside
	Type A, FD (*)	Inside/front	Axial machining – inside
	*: Define the recess type		

Machining sequence for undercuts

Main machining	Submachining	Location	Description
Undercuts			**Contour analysis/machining:** Determining the "Undercuts" form elements: ■ Type H – Machining using single paths of traverse; copying tool (type 22x) ■ Type K – Machining using single paths of traverse; copying tool (type 22x) ■ Type U – Machining using single paths of traverse; recessing tool (type 15x) ■ Type G – Machining using cycle G860 **Sequence**: First outside, then inside machining; first radial, then axial machining
	–	–	All recess types; outside and inside
	Type H, K, U, G (*)	Outside	Machining – outside
	Type H, K, U, G (*)	Inside	Machining – inside
	*: Define the undercut type		



TURN PLUS uses roughing and finishing operations to machine undercut type G. Undercut type G is only machined with an undercut cycle if no suitable roughing/ finishing tool is available.

Machining sequence for thread cutting

Main machining	Submachining	Location	Description
Thread cutting			**Contour analysis:** Determining the "Thread" form elements. **Sequence**: First outside, then inside machining; the elements are then machined according to the sequence of geometrical definition.
	–	–	Machining cylindric (longitudinal), tapered and transverse threads on the outside and inside of a contour.
	Cylindric (longitudinal), tapered, transverse (*)	Outside	Machining an external thread
	Cylindric (longitudinal), tapered, transverse (*)	Inside	Machining a threaded hole
	*: Define the type of thread		

Machining sequence for drilling

Main machining	Submachining	Location	Execution
Drilling			**Contour analysis:** Determining the "Hole" form elements. **Sequence – Drilling operations/drilling combinations:** ■ Centering / Centering and countersinking ■ Drilling ■ Countersinking / Drilling and countersinking ■ Reaming / Drilling with reaming ■ Tapping / Drilling with thread **Sequence – Location:** ■ Centric ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	–	–	Machining all bore holes on all machining locations
	Centering, drilling, countersinking, reaming, tapping (*)	–	Execution of the selected drilling operation on all machining locations
	Centering, drilling, countersinking, reaming, tapping (*)	Location	Machining a bore hole on the selected machining location
	*: Define the drilling cycle		



Drilling combinations:

- Define the drilling combinations as machining attributes (see "Machining attributes – Drilling combinations" on page 475).
- Select the appropriate drilling operation as submachining operation (see above).

Machining sequence for milling

Main machining	Submachining	Location	Execution
Milling			**Contour analysis:** Determining the milling contours. **Sequence – Milling operation:** ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	–	–	Execution of all milling operations on all machining locations
	Surface, contour, slot, pocket (*)	–	Execution of the selected milling operation on all machining locations
	Surface, contour, slot, pocket (*)	Location	Execution of the selected milling operation on the selected machining location
	*: Define the type of contour		

Machining sequence for deburring

Main machining	Submachining	Location	Execution
Deburring			**Contour analysis:** Determining milling contours with “Deburring” attribute. **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	–	–	Machining all milling contours with the attribute “Deburring” on all machining locations
	Contour, slot, pocket (*)	Location	Machining all milling contours with the attribute “Deburring” on the selected machining location
	*: Define the type of contour		

Machining sequence for engraving

Main machining	Submachining	Location	Execution
Engraving			**Contour analysis:** Determining milling contours with "Engraving" attribute. **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	–	–	Machining all milling contours with the attribute "Engraving" on all machining locations
	Contour, slot (*)	Location	Machining all milling contours with the attribute "Engraving" on the selected machining location
	*: Define the type of contour		

Machining sequence for finish-milling

Main machining	Submachining	Location	Execution
Finish-milling			**Contour analysis:** Determining the milling contours. **Sequence – Milling operation:** ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence – Location:** ■ Front (also machines Y front) ■ Lateral surface (also machines Y surface) – then the elements are machined according to the sequence of geometrical definition
	–	–	Execution of all milling contours on all machining locations
	Contour, slot, pocket (*)	Location	Execution of all milling contours on the selected machining location
	Contour, slot, pocket (*)	Location	Execution of all milling contours on the selected machining location
	*: Define the milling operation		

Machining sequence for parting and rechucking

Main machining	Submachining	Location	Execution
Parting	–	–	The workpiece is cut off.
	Full-surface machining	–	The workpiece is cut off and transferred to the opposing spindle.
Rechucking	Full-surface machining	–	■ Machine with opposing spindle: The workpiece is transferred to the opposing spindle. ■ Machine with one spindle: The workpiece is rechucked manually.

6.16 Control Graphics

During **contour definition**, TURN PLUS displays all contour elements that can be displayed.

The **IWG** and **AWG** permanently display the finished part contour and graphically depict the cutting operations. The workpiece blank **takes on a contour** during machining.

Adjusting the section (zoom function)

The zoom function can be used to isolate a detail and magnify it.

Zoom settings by keyboard:



▶ Activate the magnify function The red rectangle indicates the new section.



If several simulation windows are open:
▶ Choose the window.

▶ Adjust the section:

- To magnify, use the PgDn key
- To reduce, use the PgUp key
- To move the frame, use the arrow keys

ESC

▶ Exit the zoom function. The new section is displayed.

Zoom settings by touch pad:

▶ Position the cursor to one corner of the section.

▶ While holding the left mouse key, drag the cursor to the opposite corner of the section.

▶ Right mouse key: Return to standard size

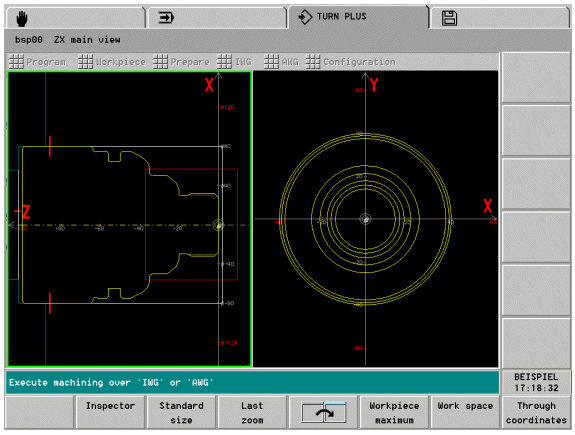
ESC

▶ Exit the zoom function. The new section is displayed.

You can make the standard setting by soft key (see table). In the “By coordinates” setting you define the magnification of the simulation window and position of the workpiece zero point.



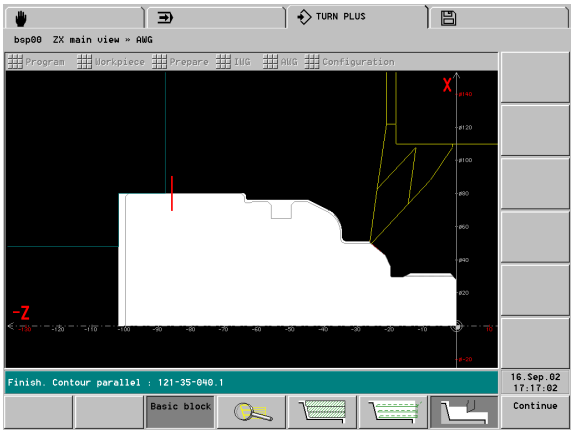
After having enlarged a detail to a great extent, select “Workpiece maximum” or “Work space,” and then isolate a new detail.



Soft keys for standard settings	
Standard size	Last setting was “workpiece maximum” or “working space”
Prev. zoom	Cancels the last magnification
Workpiece maximum	Depicts workpiece as large as possible
Work space	Depicts the work space including the tool change position
By coordinates	Adjusts the simulation window

Setting the control graphics

You can select the display of the **tool paths** and the **simulation mode** in the configuration (see "Configuring the control graphics" on page 550) or by soft key.



Window size

If several windows are displayed on screen:

- Press the "." key. The control graphics toggles between full-screen and tiled display.

Operation

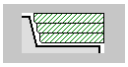
Basic block

- Soft key enabled: TURN PLUS stops after each path of traverse

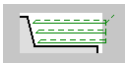
Continue

- Perform next path of traverse

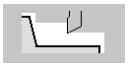
Display of the paths of traverse



- Cutting path: Depicts the surface traversed by the "cutting area" of the tool with hatch marks.



- Line display: Depicts paths of traverse with solid line (reference: theoretical tool tip).



- Material removal graphics: "Cuts" (erases) the area traversed by the "cutting area" of the tool.

6.17 Configuring TURN PLUS

With the “configuration” you change and manage the display and input variants.

General settings

Selection:

- Select “Configuration > Change.”
- Press “Settings.” TURN PLUS opens the “Settings” dialog box.

“Settings” dialog box

- Zoom behavior:
  - Dynamic: Adapts the contour graphics to the window size.
  - Static: Adapts the contour graphics to the window size when the contour is loaded; this setting is maintained.
- Level ID (designations of the coordinate axes):
  - Display
  - Do not display
- Point grid in background:
  - Display
  - Do not display
- X value input (for basic elements and form elements of the turning contour):
  - Diameter: Entries are diameter values.
  - Radius: Entries are radius values.
- With help graphic (for illustrating the input parameters):
  - Yes: Display help graphics.
  - No: Do not display help graphics.
- Start point autom.:
  - Yes: Upon calling the finished part contour entry, TURN PLUS branches to the entry of the contour starting point. The “DXF Import” soft key is not available.
  - No: After the finished part contour entry is called, you can choose whether you want to import a finished part contour or DXF contour, or whether you will enter the contour manually.



X value input: With standard forms for workpiece blank description, X values always function as diameter values. X/XE coordinates on contours for C/Y axis machining always function as radius values.

Configuring windows (views)

Define the views that TURN PLUS is to depict besides the main view (XZ plane).

Selection:

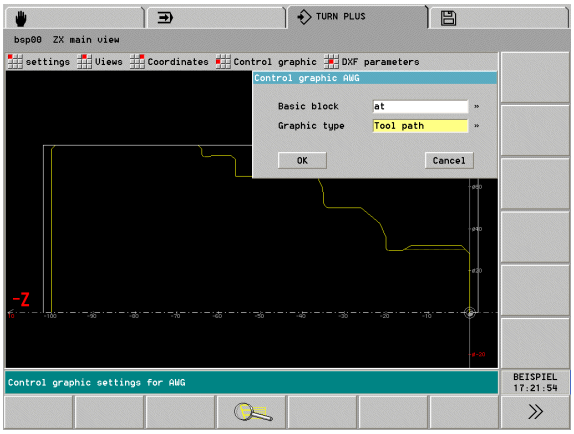
- Select “Configuration > Change.”
- Press “Views.” TURN PLUS opens the “Window configuration” dialog box.

“Window configuration” dialog box

- Views: Display of the selected views
- Selection: Select the views you want to display
- Mirror main view?
  - Yes: Display the complete contour
  - No: Display the contour above the turning center

Configuring the control graphics

With these configuration functions you edit the sequence of the control graphics and the display of the paths of traverse.



Selection:

- Select “Configuration > Change.”
- Select “Control graphic > IWG” (or “.. > AWG”). TURN PLUS opens the “Control graphic IWG/AWG” dialog box.

“Control graphic IWG/AWG” dialog box

- Basic block:
  - On: The control graphic stops after each path of traverse. Press the “Continue” soft key to start the next path of traverse.
  - Off: The control graphic simulates the machining operation without any interruptions.
- Graphic type:
  - Tool path: The control graphic depicts paths of traverse with solid line (reference: theoretical tool tip).
  - Cutting path: The control graphic depicts the surface traversed by the “cutting area” of the tool with hatch marks. The cutting path graphic accounts for the exact geometry of the tool tip (cutting radius, cutting width, tool-tip position, etc.). This graphic display is based on the tool data.
  - Material removal graphics: The blank is displayed as a white surface which is “cut” during machining.

Setting the coordinate system

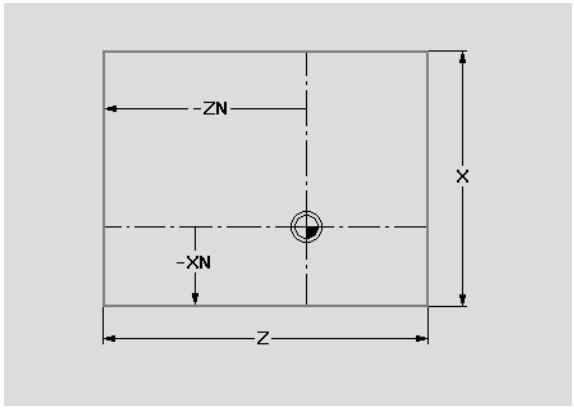
With the configuration of the "Coordinate system," you define the dimensions of the control graphics window and the position of the workpiece zero point.

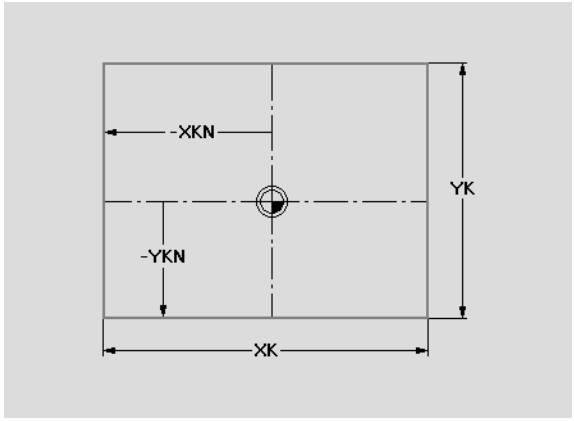
Selection:

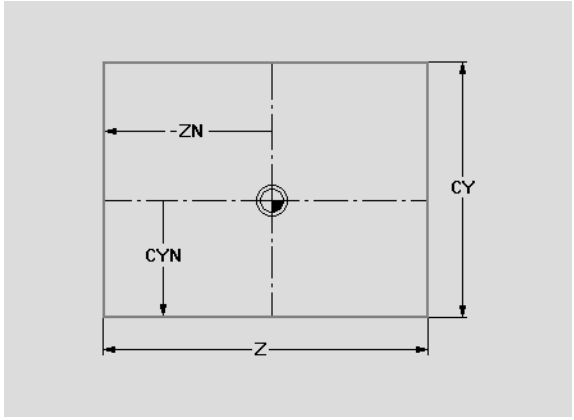
- Select "Configuration > Change."
- Select "Coordinates > Main view" (".. > Front face", ".. > Rear side" or ".. > Lateral surface"). TURN PLUS opens the "Coordinate system" dialog box.

"Coordinate system" dialog box

- For the main view (see illustration):
  - Delta X: Dimension of the control graphic window
  - Delta Z: Dimension of the control graphic window
  - XN: Position of the workpiece zero point (distance to bottom edge)
  - ZN: Position of the workpiece zero point (distance to left edge)
- For the front face (see illustration):
  - Delta YK: Dimension of the control graphic window
  - Delta XK: Dimension of the control graphic window
  - YKN: Position of the workpiece zero point (distance to bottom edge)
  - XKN: Position of the workpiece zero point (distance to left edge)
- For the rear side:
  - Delta YK: Dimension of the control graphic window
  - Delta XK: Dimension of the control graphic window
  - YKN: Position of the workpiece zero point (distance to bottom edge)
  - XKN: Position of the workpiece zero point (distance to right edge)
- For the lateral surface (see illustration):
  - Delta CY: Dimension of the control graphic window
  - Delta Z: Dimension of the control graphic window
  - CYN: Position of the workpiece zero point (distance to bottom edge)
  - ZN: Position of the workpiece zero point (distance to left edge)







TURN PLUS

- Adjusts the dimensions to the height and width of the screen.
- Increases the dimensions of the window to show the complete workpiece.

6.18 Machining Information

Tool selection, turret assignment

The **tool selection** is determined by:

- Machining direction
- Contour to be machined
- Machining sequence

If the ideal tool is not available, TURN PLUS

- First looks for a replacement tool,
- Then for an emergency tool.

If necessary, TURN PLUS adapts the machining cycle to the requirements of the replacement or emergency tool. If more than one tool is suitable for a machining operation, TURN PLUS uses the optimal tool.

TURN PLUS uses **combination tools** for drilling operations in which drilling combinations are defined.

TURN PLUS supports **multipoint tools** if they are entered in the turret list and if "From turret list" or "Combined" is defined as selection method (machining parameter 2 – WD=1 or WD=2).

Automatic turret assignment: The tool location is selected according to the "Mount type" and "Preferred tool" parameters (MP 511, ...). These parameters are used to determine whether a driven tool is supported and whether to locate primarily external/internal tools or drilling/milling tools.

The **mount type** (MP 511, ...) distinguishes between different tool mounts (see "Explanation of tool data" on page 622).



- The tool selection strategy is defined in machining parameter 2.
- TURN PLUS does not support magazine pocket systems.

Contour recessing, recess turning

The **cutting radius** must be smaller than the smallest inside radius of the recess contour, but >= 0.2 mm. TURN PLUS determines the **width of the recessing tool** from the recess contour:

- Recess contour includes paraxial base elements with radii on both sides: SB <= b + 2*r (if radii differ: smallest radius).
- Recess contour includes paraxial base elements without radii or with a radius on one side: SB <= b
- Recess contour does not include paraxial base elements: The width of the recessing tool is determined from the recessing width divisor (machining parameter 6 – SBD).

Abbreviations:

- SB: Width of recessing tool
- b: Width of base element
- r: Radius

Drilling

Depending on the geometry of the bore hole, the AWG determines the appropriate tool. For centric bore holes, TURN PLUS uses stationary tools.

Cutting data, coolant

To determine the **cutting parameters**, TURN PLUS uses the

- Workpiece material (program head)
- Cutting material (tool parameters)
- Machining mode (main machining operation selected in the IWG; main machining operation from the machining sequence in the AWG).

The values determined are multiplied by the tool-dependent compensation factors (see “Technology Database” on page 641 and see “Explanation of tool data” on page 622).

Note for roughing and finishing operations:

- Main feed rate for use of the primary cutting edge
- Auxiliary feed rate for use of the secondary cutting edge

Note for milling operations:

- Main feed rate for machining in the milling plane
- Auxiliary feed rate for infeed movements

For threading, drilling and milling operations, the cutting speed is converted into rotational speed.

Coolant: Depending on the workpiece material, cutting material and machining operation, define in the technology database whether coolant is used.

If you have specified that coolant is to be used, the AWG activates the coolant circulation for the respective machining block. If high-pressure coolant circulation is used, the AWG generates a corresponding M function.

The IWG controls coolant circulation in the same way as the AWG. Alternatively, you can define coolant circulation and pressure stage for the current machining block in the cutting data.

If you are using a "fixed" turret assignment, define the coolant circulation and the "High pressure/Normal pressure" setting separately for each tool. The AWG activates the respective coolant circulations as soon as the tool is used.

Hollowing

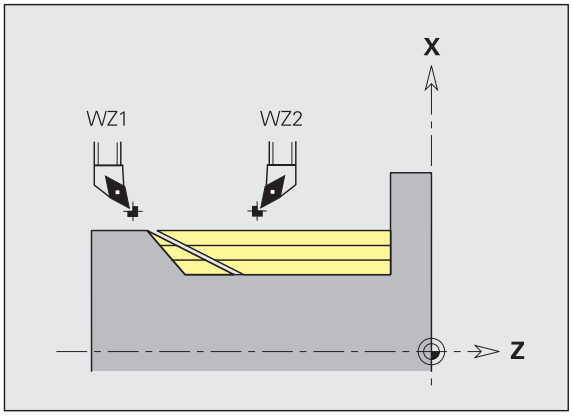
If hollowing precedes recess-turning or contour-recessing in the machining sequence, recess areas (undefined recesses) are machined with roughing tools. Otherwise, the AWG machines the respective contour areas using recessing tools. TURN PLUS uses the "inward copying angle EKW" (machining parameter 1) to distinguish recesses from relief turns.

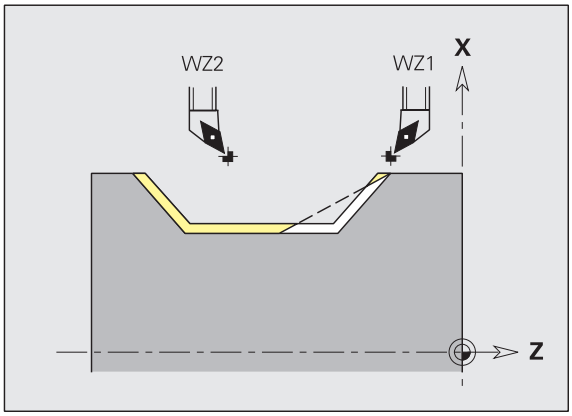
If more than one tool is required for the hollowing operation, TURN PLUS pre-machines the area with the first tool and removes the residual material with a tool machining in the opposite direction.

Contour machining (finishing): The AWG finish-machines hollowed recesses using the same strategy as for the roughing operation.

Depending on the contour and the available tools, the machining operation is executed as follows:

- Complete hollowing with one tool. If more than one tool is available, the tool with standard machining direction is used.
- If the final contour element of the area to be hollowed is a transverse element, the tool first cuts towards the transverse element (see figure).
- If the two tools have different clearance angles, the tool with the larger clearance angle is used first.
- If both tools have the same clearance angle, machining starts from the side with the smallest "inward copying angle."





Danger of collision!

During hollowing operations on the inside of contours, the plunging depth of the tool **is not checked**. Select suitable tools.

Inside contours

TURN PLUS machines continuous inside contours up to the transition from the "deepest point" to a greater diameter. The end position for drilling, roughing and finishing operations depends on:

- Cutting limit, inside
- Overhang length, inside ULI (machining parameter 4)

Prerequisite: The usable tool length must be sufficient for the machining operation. If it is not, then this parameter defines the inside machining operation. The following examples explain the principle.

Limits for internal machining operations

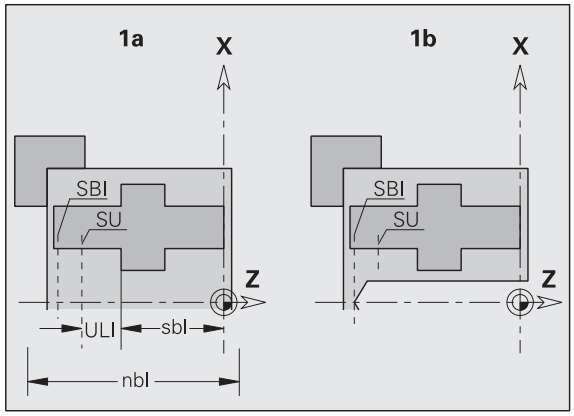
- **Predrilling: SBI** limits the drilling operation.
- **Roughing: SBI** or **SU** limit the roughing operation.
  - SU = basic length of roughing cut (sbl) + overhang length, inside (ULI)
  - To avoid residual rings during the machining process, TURN PLUS leaves an area of 5° in front of the roughing limit.
- **Finishing: sbl** limits the finishing operation.

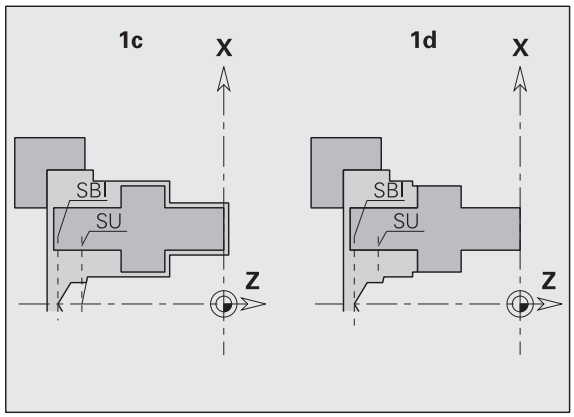
Roughing limit in front of cutting limit

Example 1: The roughing limit (SU) is located **in front of** the cutting limit, inside (SBI).

Abbreviations

- SBI: Cutting limit, inside
- SU: Roughing limit (SU = sbl + ULI)
- sbl: Basic length of roughing cut ("deepest" point of inside contour)
- ULI: Overhang length, inside (machining parameter 4)
- nbl: Usable tool length (tool parameter)



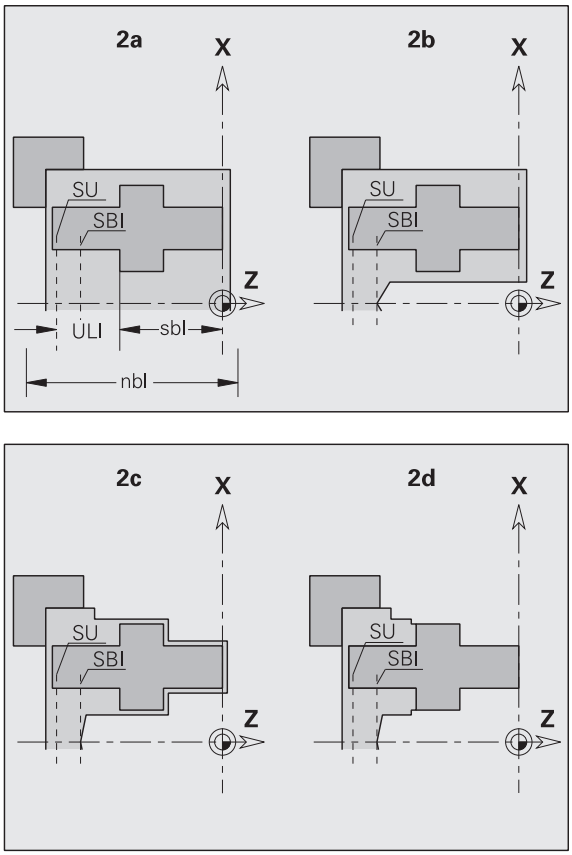


Roughing limit behind cutting limit

Example 2: The roughing limit (SU) is located **behind** the cutting limit, inside (SBI).

Abbreviations

- SBI: Cutting limit, inside
- SU: Roughing limitation (SU = sbl + ULI)
- sbl: Basic length of roughing cut ("deepest" point of inside contour)
- ULI: Overhang length, inside (machining parameter 4)
- nbl: Usable tool length (tool parameter)



Drilling

TURN PLUS distinguishes the following drilling operations:

- **Drilling without fit specifications:** The AWG selects tools that permit machining to finished dimensions. First it searches for twist drills, then for boring bars with indexable inserts.
- **Drilling with fit specifications:** The AWG drills the hole in two steps:
  - Drilling with smaller diameter than the nominal diameter of the hole.
  - "Reaming" to finished dimension



TURN PLUS only evaluates the information "with/without fit." The type of fit (H6, H7, ..) has no effect.

Shaft machining

For shafts, TURN PLUS supports rear-face machining of outside contours in addition to standard machining processes. This enables shafts to be completely machined using one setup.

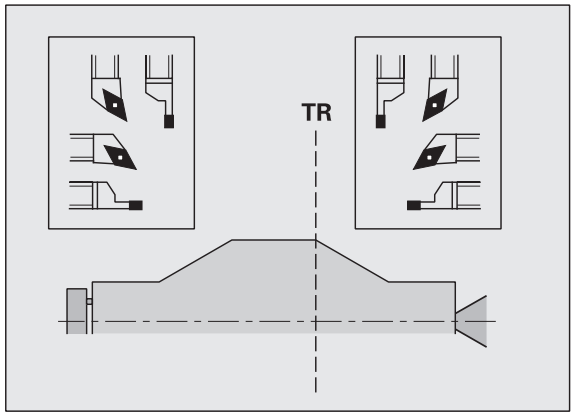
TURN PLUS does **not** support retracting the tailstock and does not check the setup used.

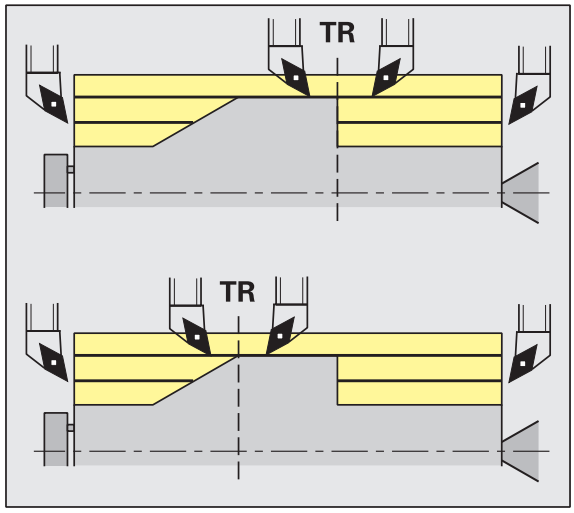
Precondition for shaft machining: The workpiece is clamped at spindle and tailstock.



Danger of collision!

TURN PLUS does not monitor for collisions during transverse machining or machining operations on the end face.





Separation point (TR)

The separation point (TR) divides the workpiece into front and rear area. If no separation point has been specified, TURN PLUS sets a separation point at the transition from the largest to a smaller diameter. Position the separation points on outside corners.

Tools for machining the

- Area on front side: Main machining direction –Z; or primarily “left” recessing or tapping tools, etc.
- Area on rear side: Main machining direction +Z; or primarily “right” recessing or tapping tools, etc.

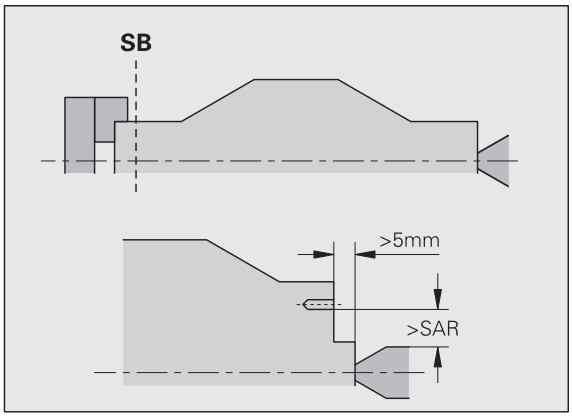
Setting/changing the separation point: see “Machining attributes – Separation point” on page 480

Protective zones for drilling and milling operations

TURN PLUS machines drilling and milling contours on transverse surfaces (front/rear face) if:

- (Horizontal) distance to transverse surface > 5 mm, or
- Distance between chucking equipment and drilling/milling contour is > SAR (SAR: See machining parameter 2).

If jaws are used for clamping the shaft at the spindle, TURN PLUS accounts for the cutting limitation (SB).



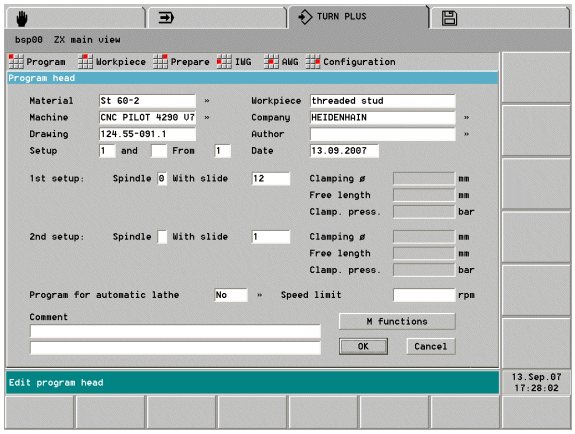
Machining information

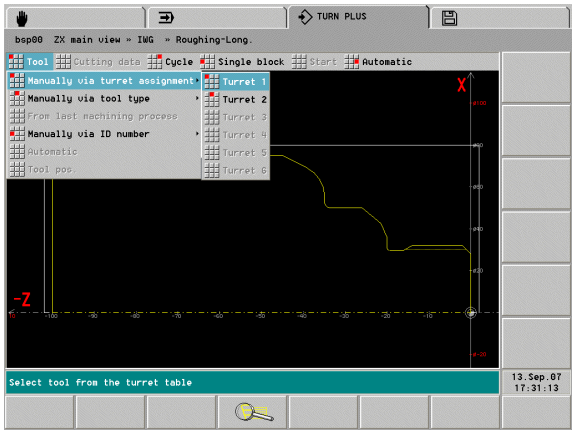
- **Chucking the workpiece at the spindle:** Ensure that the area, where the blank part is chucked, is premachined. Otherwise, the cutting limitation might adversely affect the machining strategies.
- **Machining of bars:** TURN PLUS **does not control** the bar loader and does not move the tailstock and steady rest components. TURN PLUS does not support workpiece adjustment between collet and dead center during machining operations.
- **Transverse machining**
  - Please note that the entries made in the machining sequence apply to the complete workpiece and thus also to the transverse machining of shaft ends.
  - The AWG does not machine inside areas on the rear face. If jaws are used for clamping the shaft at the spindle, the rear face is not machined.
- **Longitudinal machining:** First the front area is machined, then the rear area.
- **Collision prevention:** If machining operations are not performed **without collisions**, you can do the following:
  - Add a retraction of the tailstock, a positioning of the steady rest, etc. to the DIN PLUS program.
  - Add cutting limits to the DIN PLUS program to avoid collisions.
  - Disable automatic machining in the AWG by assigning the "Exclusion from machining" attribute or by defining a specific machining location in the machining sequence.
  - Define an oversize=0 for the workpiece blank. As a consequence, the front area is not machined (e.g. shafts cut to length and centered shafts).

Multi-slide machines

On machines with more than one slide, you can influence the tool selection and program generation through the following parameters:

- **Program head:** In the “1st setup: Spindle .. With slide ..” box, specify the slides you want to use for the machining operation. Enter the slide numbers one after the other, without separators (see illustration). This also applies to the second setup.
- **Tool selection in the IWG:** The slides or turrets defined in the program head are taken into account by the IWG. Choose the turret in which you want to insert the tool.
- **Tool selection in the AWG:** The slides or turrets defined in the program head are taken into account by the AWG. In machining parameter “Sequence of tool selection” (parameter 22), define the sequence in which the tool carriers of the slides are to be configured.





Full-surface machining

You describe the geometry of the blank and finished part, and TURN PLUS generates the working plan for the **complete workpiece.**

Preconditions for full-surface machining:

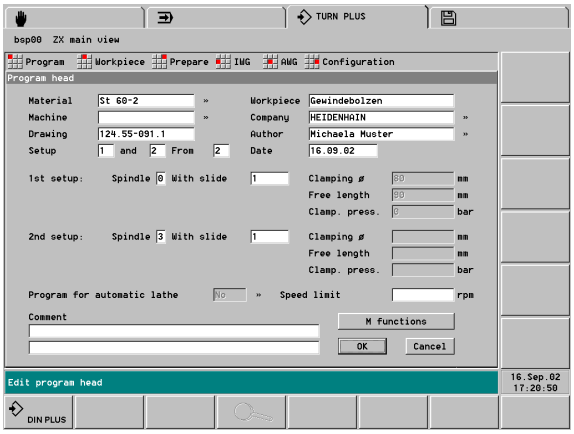
- In the **program head** the spindle and the slide for the second setup are defined ("2nd setup .." input boxes).
- In the **machining sequence** the main machining operation "Rechucking"" or "Parting" is entered after the machining of the front face.

Depending on the main machining and submachining entries defined in the machining sequence, TURN PLUS activates one of the following **expert programs** (machining parameter 21):

- "Rechucking – Full-surface machining" entry: TURN PLUS activates the expert program entered in SP-UMKOMPL. The opposing spindle takes over the workpiece.
- "Parting – Full-surface machining" entry: TURN PLUS activates the expert program entered in SP-UMKOMPLA. The workpiece is cut off and transferred to the opposing spindle (machining of bars).

To influence rear-face machining in the machining sequence: see "Machining sequence – Fundamentals" on page 535

The NC program generated includes front and back machining (including the drilling, milling and inside machining), calling the expert program, and the clamping information of both setups.



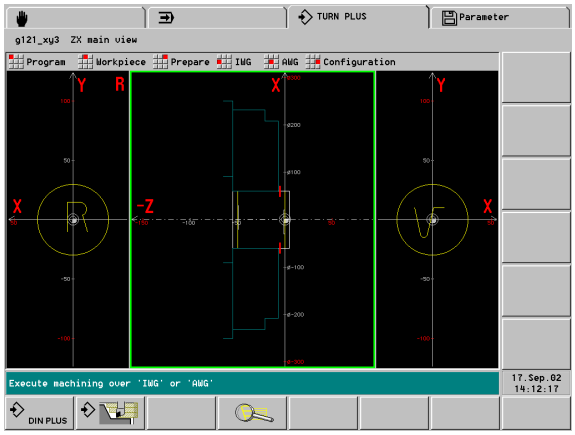
Note on machining the rear face

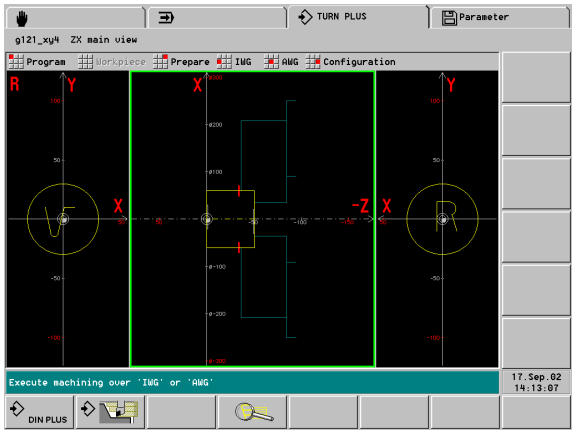
For contours on the rear side (C/Y axis machining) remember the orientation of the XK or X axis and the orientation of the C axis.

Designations (see figure at right):

- Front side (“V”): The side toward the working space
- Rear side (“R”): The side away from the working space

These designations also apply to workpieces clamped at the opposing spindle, or to workpieces rechucked for rear-side machining in lathes with one spindle.

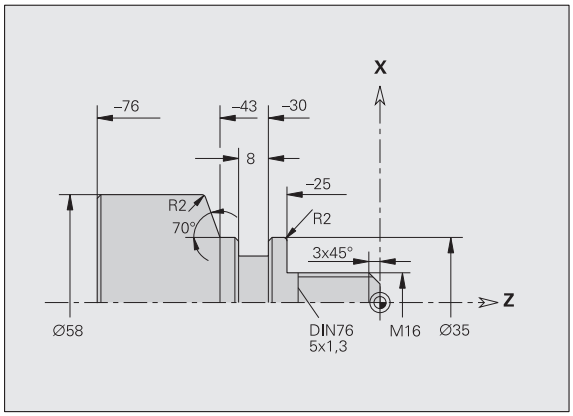




6.19 Example

On the basis of the production drawing, the working steps for defining the contour of the blank and finished part, the setup procedures and automatic working plan generation are explained.

Workpiece blank: Ø60 X 80; Material: Ck 45



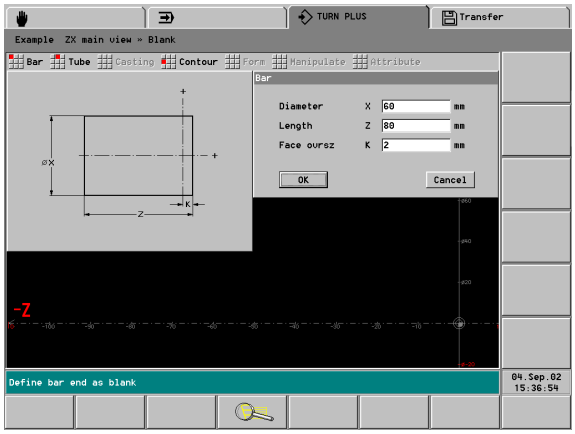
- Undefined chamfers: 1x45°
- Undefined radii: 1 mm

Creating a program

- Select "Program > New". TURN PLUS opens the "New program" dialog box.
- Inputs:
  - Program name
  - Material (select from fixed-word list)
- Click on the Program Head button. TURN PLUS opens the "Program head" dialog box.
- Inputs:
  - "Spindle – Slide for 1st setup"
  - Make further entries, as required.
- Return to the "New program" dialog box.
- TURN PLUS creates the new program.

Defining the workpiece blank

- Select “Workpiece > Blank > Bar.” TURN PLUS opens the “Bar” dialog box.
- Inputs:
  - Diameter = 60 mm
  - Length = 80 mm
  - Oversize = 2 mm
- TURN PLUS displays the workpiece blank.



ESC

- Press the ESC key to return to the main menu.

Defining the basic contour

- Select “Workpiece > Finished part (> Contour).”
- “Point” dialog box (starting point of contour):
  - Enter X = 0; Z = 0





- Enter X = 16



- Enter Z = –25



- Enter X = 35



- Enter Z = –43



- Enter X = 58; W = 70



- Enter Z = –76

ESC

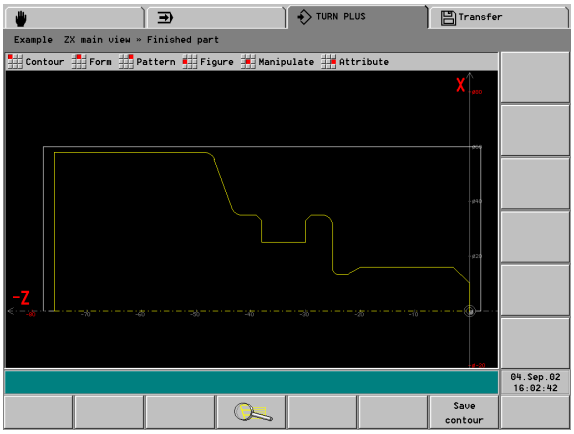
- Press the ESC key to go back one menu level.

ESC

- Press the ESC key. TURN PLUS asks whether you want to close the contour.

- Press “Yes.” TURN PLUS creates the basic contour.

Defining form elements



Chamfer at corner of threaded shank:

- Select “Form > Chamfer.”
- Select the corner of the threaded shank.
- “Chamfer” dialog box: Chamfer width = 3 mm

Rounding arcs:

- Select “Form > Rounding.”
- Select the corners for the rounding arcs.
- “Rounding” dialog box: Rounding radius = 2 mm

Undercut:

- Select “Form > Undercut > Undercut type G.”
- Select the corner for the undercut.
- “Undercut type G” dialog box:
  - Undercut length = 5 mm
  - Undercut depth = 1.3 mm
  - Approach angle = 30 °

Recess:

- Select “Form > Recess > Recess type D.”
- Select the basic element for the recess.
- “Recess type D” dialog box:
  - Reference point (Z) = –30 mm
  - Recess width (Ki) = –8 mm
  - Recess diameter = 25 mm
  - Corners (B): Chamfers; 1 mm

Thread:

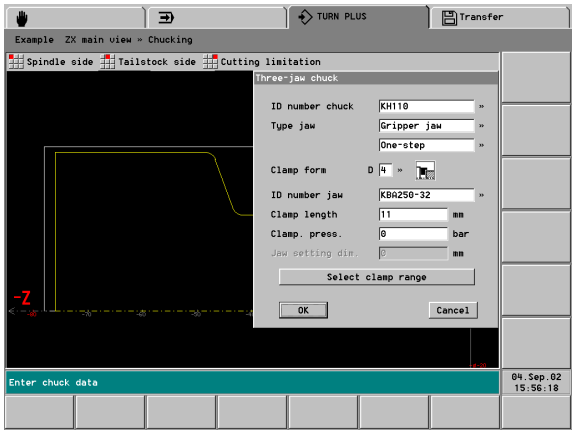
- Select “Form > Thread.”
- Select the basic element for the thread.
- “Thread” dialog box: Select “metric ISO thread.”

ESC

- Press the ESC key to return to the main menu.

Preparing the machining process, chucking

- Select “Prepare > Chucking > Clamp.”
- Select “Spindle side > Three-jaw chuck.”
- “Three-jaw chuck” dialog box:
  - Select the “ID number chuck.”
  - Enter the type of jaw.
  - Enter the clamp form.
  - Select the “ID number jaw.”
  - Check/enter the clamp length and clamping pressure.
  - Define the clamp range: Select a contour element touched by the jaws.
- TURN PLUS displays the selected chucking equipment and the cutting limit.





- Press the ESC key to return to the main menu.

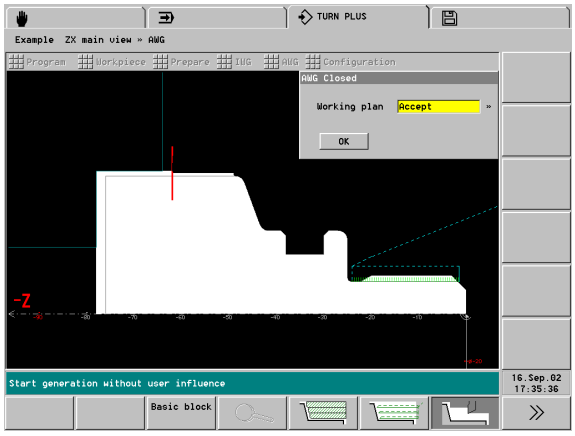
Generating and saving a working plan

Generating a working plan

- Select “AWG > Automatic.”
- TURN PLUS runs a graphic simulation of the machining process.
- Select “Accept working plan.”

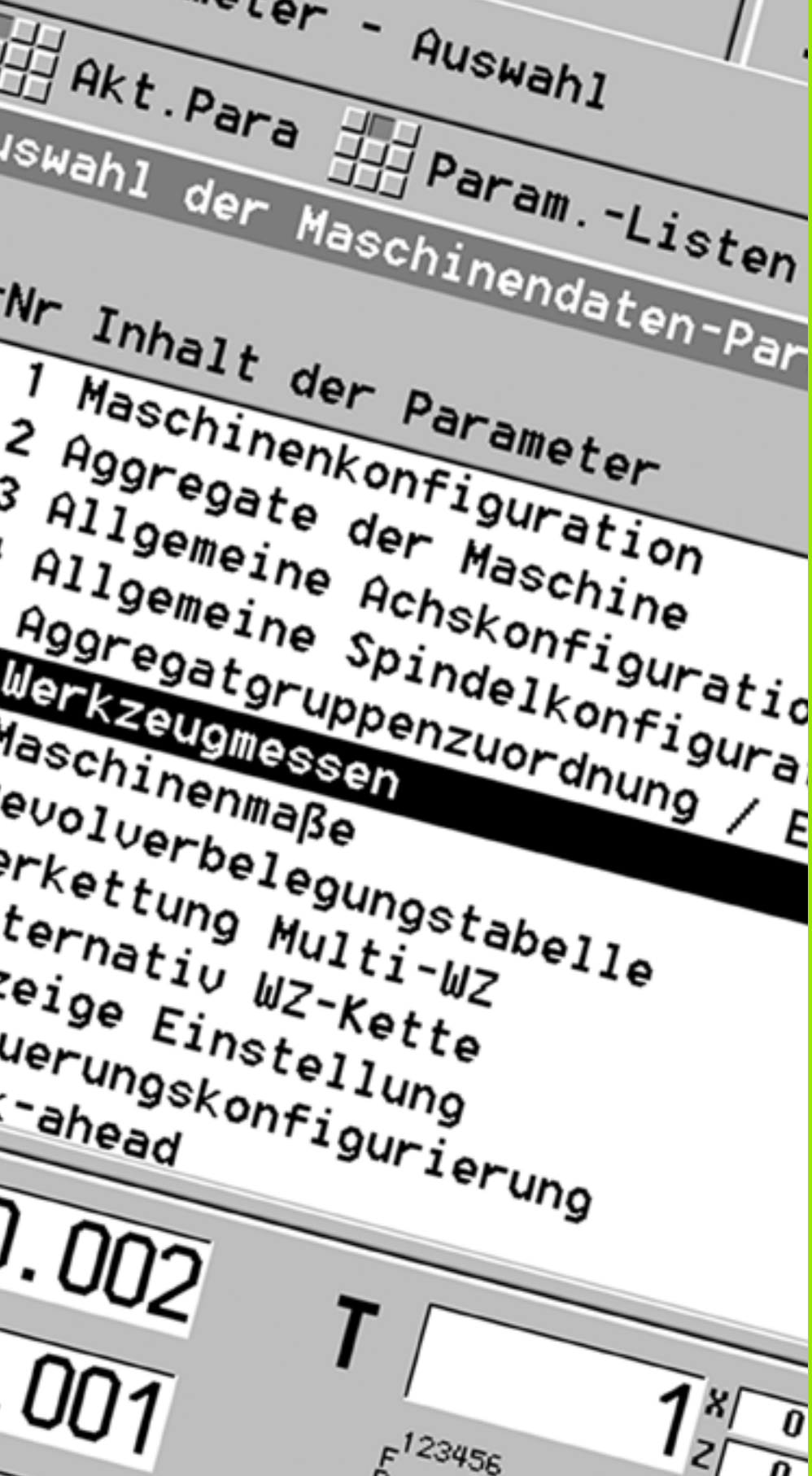
Saving the program

- Select “Program > Save > Complete.”
- Check/edit the file name.
- TURN PLUS saves:
  - the working plan, the blank contour and the finished part contour (in a file)
  - the NC program (DIN PLUS format).





The AWG generates the work blocks according to the machining sequence and the settings of the machining parameters.



7

Parameters

7.1 The Parameter Mode of Operation

The parameters of the CNC PILOT are grouped as follows:

- **Machine parameters:** These are used to adapt the control to the requirements of the machine (e.g. parameters for components, assemblies, the assignment of axes, slides and spindles, etc.).
- **Control parameters:** These are used to configure the control system (machine display, interfaces, measuring system used, etc.).
- **Setup parameters:** These define certain settings required for the production of a specific part (workpiece zero point, tool change position, compensation values, etc.).
- **PLC parameters:** These parameters are defined by the machine manufacturer (see Machine Manual).
- **Machining parameters:** These define the strategy for machining cycles and for TURN PLUS.

In this operating mode, the following operating resource and technology parameters are managed in addition:

- Tool parameters
- Chucking equipment parameters
- Technology parameters (cutting values)

This Manual describes parameters that can be changed by the user ("system manager" user class). Other parameters are explained in the Technical Manual.

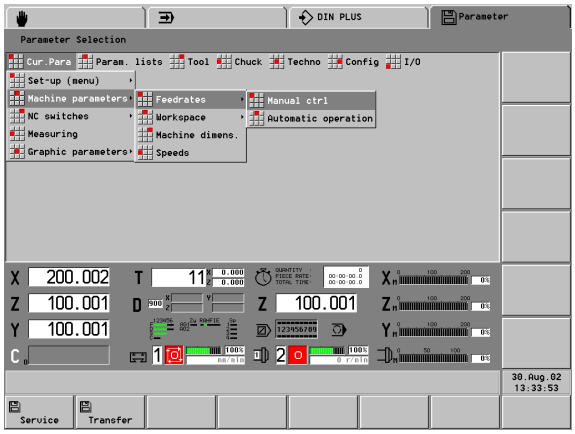
Data exchange and data backup: The CNC PILOT supports data exchange of parameters and associated fixed-word lists. All parameters are included for data backup.

To transfer and back up data, you use the Transfer mode of operation (see "Transmitting parameters/operating resources" on page 675).

7.2 Editing Parameters

Current parameters

This drop-down menu presents an overview of frequently used parameters so that you can select them without having to know the parameter number.



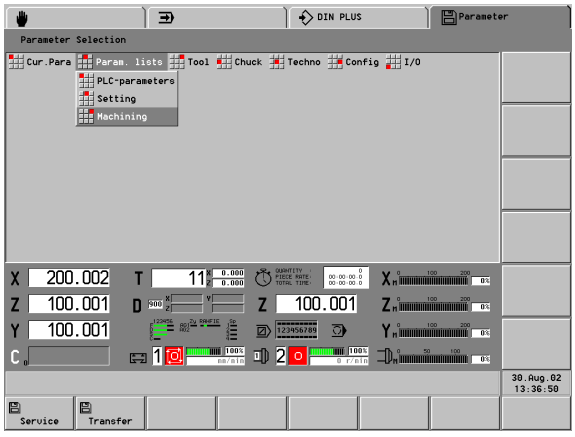
Edit parameter

- If required, log on as system manager (Service mode).
- Select the parameter from the “Cur.Para > ..” drop-down menu. The CNC PILOT presents the parameter for editing.
- Make the required changes and close the dialog box.

Parameter lists

The following parameter groups are provided in the “Param. lists” drop-down menu; you can select these parameters without being logged on:

- PLC Parameters
- Set-up parameters
- Machining parameters



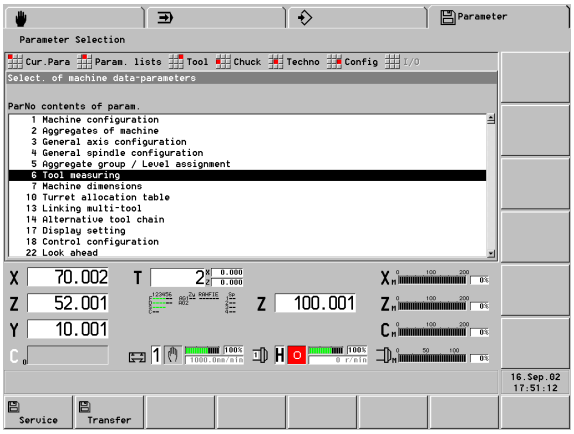
Editing set-up/machining parameters

- Select „Param. lists > Machining” (“.. > Setting” or “.. > PLC-parameters”). The CNC PILOT opens the corresponding parameter list.
- Select the parameter.
- Press ENTER. The CNC PILOT presents the parameter for editing.
- Make the changes

Editing configuration parameters

In the “Config” drop-down menu, all parameter groups are presented for selection. The procedure is identical with that described in this section.

- The CNC PILOT checks whether the user is authorized to change a parameter. Log on as system manager if you wish to edit protected parameters. Otherwise, you are only authorized to read parameters.
- Parameters that influence the production of a workpiece cannot be edited in automatic mode.



Editing configuration parameters

Log on as system manager (Service mode).

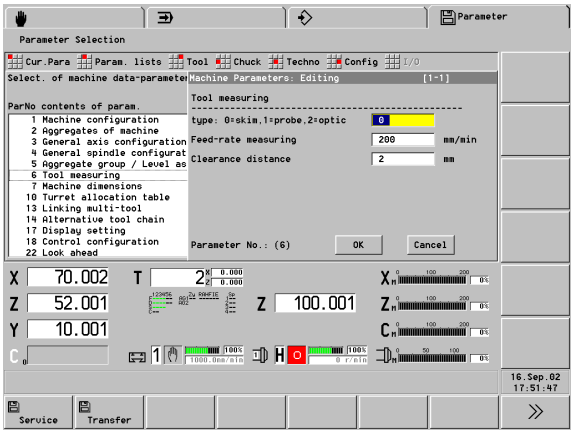
Parameter number is not known:

Select “Config > Machine list” (or “.. > Control list”).

Select the parameter.

Press ENTER. The CNC PILOT presents the parameter for editing.

Make the changes



Parameter number is known:

Select “Config > Machine direct” (or “.. > Control direct”).

Enter the parameter number and call the parameter. The CNC PILOT presents the parameter for editing.

7.3 Machine Parameters (MP)

Value ranges of machine parameters:

- 1..200: General machine configuration
- 201..500: Slides 1 to 6 (50 positions per slide)
- 501..800: Tool carriers 1 to 6 (50 positions per tool carrier)
- 801..1000: Spindles 1 to 4 (50 positions per spindle)
- 1001..1100: C-axes 1 to 2 (50 positions per C axis)
- 1101..2000: Axes 1 to 16 (50 positions per axis)
- 2001..2100: Various components of the machine (these parameters are not used at present)

General machine parameters

General machine parameters	
6	**Tool measurement** These parameters define how to determine the tool lengths in set-up mode. ■ **Type** of tool measurement: ■ 0: Touch-off ■ 1: Touch probe ■ 2: Optical gauge ■ **Measuring feed:** Feed rate for approaching the touch probe ■ **Clearance distance:** Minimum distance, which has to be traversed in the opposite direction to the measuring direction, to retract the touch probe after the stylus has deflected.
7	**Machine dimensions** Within the framework of variable programming, machine dimensions can be used in NC programs. The use of the machine dimensions is defined only by the NC program. ■ Dimension n X, Y, Z, U, V, W, A, B, C (n: 1 to 9)
17	**Display settings** The "Display mode" defines the position display (actual value display) within the machine display. ■ **Actual position display type** ■ 0: Actual value ■ 1: Following error ■ 2: Distance of traverse ■ 3: Tool point — with reference to machine datum ■ 4: Slide position ■ 5: Distance from reference dog to zero pulse ■ 6: Nominal position ■ 7: Difference from tool point to slide position ■ 8: Nominal director position

General machine parameters

18 **Control configuration**

- **PLC is to perform counting of workpieces:**
  - 0: CNC is to perform counting of workpieces
  - 1: PLC is to perform counting of workpieces
- **M0/M1 for all NC channels**
  - 0: M0/M1 triggers a STOP on programmed channels.
  - 1: M0/M1 triggers a STOP on all channels.
- **Interpreter stop upon tool change**
  - 0: No interpreter stop
  - 1: Interpreter stop. Look-ahead block interpretation is stopped, and reactivated after the T command has been processed.

Machine parameters for slides

Slide parameters

204, 254, .. **Feed rates**

Rapid traverse speed and feed rate when slide is moved with the axis-direction keys (jog keys).

- **Rapid traverse contouring speed for manual control**
- **Feed rate contouring speed for manual control**

205, 255, .. **Protection zone monitoring**

The protection zone dimensions are defined for individual axes (MP 1116, ...). Define in this parameter whether the protection zone dimensions are to be monitored.

- **Monitoring**
  - 0: Protection zone monitoring OFF
  - 1: Protection zone monitoring ON

The other parameters are not used at present.

208, 258, .. **Thread cutting**

If the "coupling/uncoupling path" is not programmed in an NC program, parameter values are used.

- **Coupling path:** Acceleration path at the beginning of a threading cut in order to synchronize the feed axis with the spindle.
- **Decoupling path:** Deceleration path at the end of a threading cut.

209, 259, .. **Slide switch-off**

- **Slide**
  - 0: Switch off the slide
  - 1: Do not switch off the slide

Slide parameters	
211, 261, ..	**Position of the touch probe or optical gauge** To define the position of the probe, enter its external coordinates (reference: machine zero point). To define the position of the optical measuring system, enter the position of the cross hairs (+X/+Z). ■ **Position of touch probe/optical gauge in +X** ■ **Position of touch probe in –X** ■ **Position of touch probe/optical gauge in +Z** ■ **Position of touch probe in –Z**
511..542, 561..592, ..	**Description of tool holders** These parameters describe the positions of the tool holders relative to the tool-carrier reference point. ■ **Distance to carrier reference point X/Z/Y:** Distance between reference point of tool carrier and reference point of tool holder. ■ **Compensation X/Z/Y:** Compensation value for the distance between reference point of tool carrier and reference point of tool holder.

Machine parameters for spindles

Spindle parameters	
804, 854, ..	**Spindle protection zone monitoring** is not used at present.
805, 855, ..	**General spindle parameters** ■ **Zero point shift (M19):** Defines the distance between the reference point of the spindle and the reference point of the encoder. After receiving the reference pulse from the encoder, the value determined is transferred. ■ **Number of revolutions for backing off:** Number of spindle revolutions after the spindle has stopped in Automatic mode. (When using low spindle speeds, additional spindle revolutions are necessary to relieve the tool.)
806, 856, ..	**Tolerance values of spindle** ■ **Speed tolerance [%]:** The system proceeds from a G0 block to a G1 block as soon as the status "Speed reached" is attained. When the speed lies within the tolerance range, the status is attained. The tolerance value refers to the nominal value. ■ **Position window, position [°]:** After the spindle has stopped at a defined position (M19), the system proceeds to the next block as soon as the status "Position reached" is attained. When the difference between nominal and actual value lies within the tolerance range, the status is attained. The tolerance value refers to the nominal value. ■ **Speed tolerance, synchronization [rpm]:**:Criterion for status "Synchronization reached." ■ **Position tolerance, synchronization [°]:** Criterion for status "Synchronization reached." Notes on synchronization parameters: ■ The synchronization parameters depend on the settings of the slave spindle. ■ The "Synchronization reached" status is attained when the actual speed difference and the actual position difference between the spindles to be synchronized lies within the tolerance range. When the status "Synchronization reached" is attained, the torque of the guided spindle is limited. ■ The tolerances which are actually attainable on the machine must not be undercut when setting these parameters. The value programmed here must be set to a value greater than the sum of the maximum differences in the speeds of the guiding and guided spindles (approx. 5 to 10 rpm).

Spindle parameters	
807, 857, ..	**Measuring the angle offset (G906) for spindle** Evaluation: G906 Measuring angular offset during spindle synchronization ■ **Maximum permissible change in position:** Tolerance window for the change of position offset after the spindles have gripped the workpiece at both ends during synchronized operation. If the offset change exceeds the maximum value, an error message appears. A normal fluctuation of approx. 0.5° must be considered. ■ **Waiting time, measuring offset:** Measuring period (duration)
808, 858, ..	**Parting control (G991) for spindle** Evaluation: G991 Controlled parting using spindle monitoring After the parting operation has been completed, the phase angle of the two synchronized spindles changes without the nominal values (speed/angle of rotation) being changed. If the speed difference is exceeded during the monitoring time, the workpiece is considered as being cut off. ■ **Speed difference** ■ **Monitoring time**
809, 859, ..	**Load monitoring for spindle** Evaluation: Load monitoring ■ **Start-up time for monitoring [0..1000 ms]:** The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed. The parameter is only evaluated if "Omit paths of rapid traverse" is active. ■ **Number of measured values to be averaged [1..50]:** The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring. ■ **Reaction delay time P1, P2 [0..1000 ms]:** The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded. ■ **Maximum torque** is not used at present.

Machine parameters for C axes

Parameters for C axes	
1007, 1057, ..	**Backlash compensation in C axis** During each change in direction, the nominal value is corrected by the value entered in the "Backlash compensation value" option. ■ **Type of backlash compensation** ■ 0: No backlash compensation ■ 1: During a change in direction, the value of the backlash compensation is added. ■ **Value of backlash compensation**

Parameters for C axes	
1010, 1060, ..	**Load monitoring for C axis** Evaluation: Load monitoring ■ **Start-up time for monitoring [0..1000 ms]:** The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed. The parameter is only evaluated if "Omit paths of rapid traverse" is active. ■ **Number of measured values to be averaged [1..50]:** The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring. ■ **Maximum torque** is not used at present. ■ **Reaction delay time P1, P2 [0..1000 ms]:** The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded.
1016, 1066, ..	**Limit switches and rapid-traverse speed in C axis** ■ **Rapid-traverse speed, C axis:** Maximum speed for spindle positioning
1019, 1069, ..	**General data, C axis** This parameter is evaluated if "pre-positioning" is switched on ("configuration code 1" – MP 18). For digital drives, pre-positioning is usually not necessary. ■ **Pre-positioning of spindle with M14:** Angle at which the spindle is positioned before the C axis is swiveled in.
1020, 1070, ..	**Angle compensation in C axis:** Parameters are entered by the machine tool builder.
1021..1026, 1071..1076, ..	**Compensation values in C axis:** Parameters are entered by the machine tool builder.

Machine parameters for linear axes

Parameters for linear axes	
1107, 1157, ..	**Backlash compensation for linear axis** During each change in direction, the nominal value is corrected by the value entered in the "Backlash compensation value" option. ■ **Type of backlash compensation** ■ 0: No backlash compensation ■ 1: During a change in direction, the value of the backlash compensation is added. ■ **Value of backlash compensation**

Parameters for linear axes	
1110, 1160, ..	**Load monitoring for linear axis** Evaluation: Load monitoring ■ **Start-up time for monitoring [0..1000 ms]:** The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed. Evaluated if "Omit paths of rapid traverse" is active. ■ **Number of measured values to be averaged [1..50]:** The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring. ■ **Maximum torque** is not used at present. ■ **Reaction delay time P1, P2 [0..1000 ms]:** The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded.
1112, 1162, ..	**Traversing to a fixed stop (G916) in linear axis** Evaluation: G916 Traverse to fixed stop These parameters apply to the linear axis for which G916 has been programmed. ■ **Lag error limit:** The slide is stopped as soon as the following error (lag error; deviation of the actual from the nominal position) reaches the following error limit. ■ **Reversal distance:** After reaching the "fixed stop," the slide is moved back by the reversal distance (to reduce the stress).
1114, 1164, ..	**Zero point offset during conversion in linear axis** ■ **NC zero offset:** Length by which the machine zero point is shifted during conversion (G30).
1115, 1165, ..	**Parting control (G917) for linear axis** These parameters apply to the linear axis for which G917 has been programmed. Evaluation: G917 Controlled parting using following error (lag error) monitoring ■ **Lag error limit:** The slide is stopped as soon as the difference between actual and nominal position has reached the following error limit. The CNC PILOT reports that it has detected following error. ■ **Feed rate** when moving the linear axis using following error monitoring.
1116, 1166, ..	**Limit switches, protection zone, feed rates in linear axis** ■ **Negative dimension of protection zone** ■ **Positive dimension for protection zone:** Dimensions for protection zone monitoring (reference: machine zero point) ■ **Rapid-traverse speed** in Automatic mode ■ **Reference dimension:** Distance between reference point and machine zero point
1120, 1170, ..	**Misalignment compensation for linear axis:** Parameters are entered by the machine tool builder.

7.4 Control Parameters

General control parameters

General control parameters	
1	**Settings** ■ Suppressing **printer output:** With the PRINTA command in the NC program you can output data to a printer (see also control parameter 40). ■ 0: Suppress the output ■ 1: Enable the output ■ **Metric / Inches:** Setting for the unit of measure. ■ 0: Metric ■ 1: Inches ■ **Format of the position displays** (actual value displays). ■ 0: Format 4.3 (4 integral, 3 decimal places) ■ 1: Format 3.4 (3 integral, 4 decimal places) **Notes:** ■ With DIN PLUS programs, the unit of measure defined in the program head is decisive — regardless of the unit of measure set here. ■ Restart the CNC PILOT whenever you have changed the unit of measure.
8	**Settings for load monitoring** Calculation of limit values: Limit value = reference value * factor for limit value Evaluation: Load monitoring ■ **Factor of torque limit 1** ■ **Factor of torque limit 2** ■ **Factor of working limit value** ■ **Minimum torque** [% of rated torque]: Reference values below this value are raised to this minimum torque value. This prevents that limit values are exceeded as a result of minor torque differences. ■ **Maximum file size [KB]:** If the data during measured value registration exceed the maximum file size, the "oldest" values are overwritten. Rule of thumb: One component requires approx. 12 KB per minute of program run time.
10	**Post-process measuring** Evaluation: Post-process measuring ■ **Switching measurement on/off** ■ 0: Post-process measurement off ■ 1: Post-process measurement on. The CNC PILOT is ready to receive data. ■ **Type of measurement** ■ 1: Post-process measuring ■ **Measured value coupling** ■ 0: New measured values overwrite old measured values. ■ 1: New measured values are not received until the old measured values are evaluated. Note: In control parameters 40, ..., the serial interface is selected and the interface parameters are defined.

General control parameters	
11	**FTP parameters** Evaluation: Data transfer using FTP (file transfer protocol) ■ **User name:** Name of one's own station ■ **Passcode** ■ **Address/name of FTP server:** Address/name of communications partner ■ **Use FTP** ■ 0: No ■ 1: Yes Note: The parameters can also be set using the transfer functions.
40	**Assignment to the interfaces** The interface parameters are saved in the parameters 41 to 47. In parameter 40 the machine tool builder assigns an interface description to an encoder. The Transfer mode of operation uses the parameters of the interface defined under "external input/output." Meaning of the entries: ■ 1..7: Interface 1..7 – Example: "2 = interface 2" (control parameter 42) ■ External input/output ■ DATAPILOT 90 ■ Printer ■ Post-process measuring ■ 2. 2nd keyboard (or card reader) Note: The parameter settings are made by the machine tool builder.
41..47	**Interfaces** The CNC PILOT stores the "settings" of the serial interfaces and the printer interface in these parameters. Note: You define the parameter settings in the Transfer mode of operation.
48	**Transfer directory** ■ **NETWORK directory:** Path of directory offered and indicated for data transfer with NETWORK. Note: You define the parameter settings in the Transfer mode of operation.

Control parameters for simulation

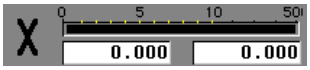
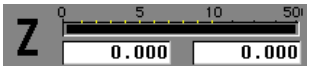
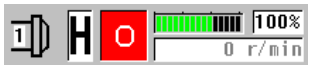
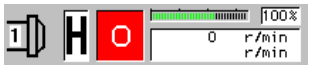
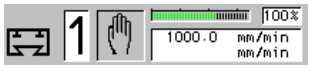
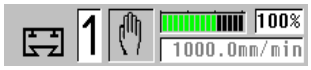
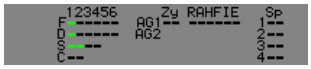
Parameters for simulation	
20	**Time counting for general simulation** These times are used as nonmachining times for the time calculation function. Evaluation: Time calculation (Simulation mode) ■ **Tool change time [sec]** ■ **Gear shifting time [sec]** ■ **Time allowance for M functions [sec]:** All M functions are rated with this time. If an M function is also specified in control parameter 21, the time allowances entered here are added.
21	**Time counting for simulation: M function** Declare individual time allowances for a maximum of 10 M functions. Evaluation: Time calculation in simulation mode ■ **1..10. M function:** Number of M function ■ **Time allowance [sec]:** Individual time allowance. The time calculation function in simulation mode adds the specified time to the time allowance entered in control parameter 20.
22	**Simulation: Default window size (X, Z)** The simulation function adapts the window size to the workpiece blank. If no blank part is programmed, the CNC PILOT uses a standard window size. Evaluation: Simulation mode ■ **Zero position X:** Distance of the origin of the coordinate system from the lower window frame ■ **Zero position Z:** Distance of the origin of the coordinate system from the left window frame ■ **Delta X:** Vertical expansion of the graphics window ■ **Delta Z:** Horizontal expansion of the graphics window
23	**Simulation: Standard workpiece blank** If no blank part is programmed, the CNC PILOT uses a standard blank. Evaluation: Simulation mode ■ **Outside diameter** ■ **Workpiece blank length** ■ **Right edge of blank part** (oversize), reference: Workpiece zero point ■ **Inside diameter** for hollow cylinders, solid blanks: 0

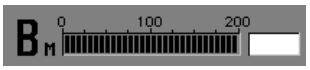
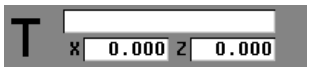
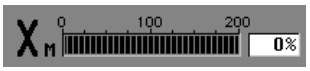
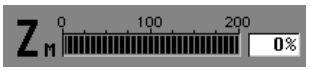
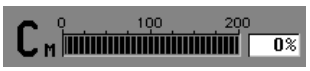
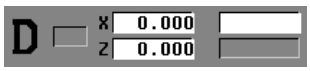
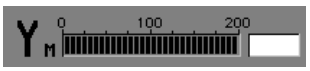
Parameters for simulation	
24	**Simulation: Color table for feed travel** The feed travel of a tool is displayed in the color assigned to the respective turret location. Evaluation: Simulation mode ■ **Color for turret position n (n: 1..16)** – Color code: ■ 0: Light green (standard color) ■ 1: Dark gray ■ 2: Light gray ■ 3: Dark blue ■ 4: Light blue ■ 5: Dark green ■ 6: Light green ■ 7: Dark red ■ 8: Light red ■ 9: Yellow ■ 10: White
27	**Simulation: Settings** Evaluation: Simulation mode ■ **Path delay (machining):** The machining simulation and the control graphics (TURN PLUS) are delayed by the "path delay" time after each path that has been simulated graphically. The simulation speed can thus be influenced. Smallest unit: 10 msec

Control parameters for machine display

Parameters for machine display	
301..306, 313..318, ..	**Display type 1..6 manual control** The machine display consists of 12 configurable fields (see tables below).
307..312, 319..324, ..	**Display type 1..6 automatic** The machine display comprises 12 configurable fields (for arrangement, see table below). ■ **Symbol, field n (n: 1..16):** Code number of the "symbol" (for code numbers, see following pages). ■ **Slide / Spindle:** Slide, spindle or C axis to be displayed. The CNC PILOT automatically distinguishes between slide, spindle and C axis. ■ 0: The component selected by slide/spindle-change key is displayed. ■ >0: Slide, spindle or C-axis number ■ **Component group:** Always enter 0.

Arrangement of the machine display fields			
Field 1	Field 5	Field 9	Field 13
Field 2	Field 6	Field 10	Field 14
Field 3	Field 7	Field 11	Field 15
Field 4	Field 8	Field 12	Field 16

Code numbers for “symbols”			Code numbers for “symbols”		
0	Special code, no display				
1	Actual X-position display		34	Distance to go in b and actual position (auxiliary axis)	
2	Actual Z-position display		35	Distance to go in c and actual position (auxiliary axis)	
3	Actual C-position display		41	Quantity and piece rate information	
4	Actual value in Y		42	Quantity information	
5	Distance to go in X and actual position		43	Piece rate information	
6	Distance to go in Z and actual position		45	M01 and skip levels	
8	Distance to go in Y and actual position		60	Spindle and speed information	
10	All principal axes		61	Actual/nominal speed	
11	All auxiliary axes		69	Actl/noml feed rate	
12	Actual U-value display (auxiliary axis)		70	Slide and feed rate information	
13	Actual V-value display (auxiliary axis)		71	Channel display	
14	Actual W-value display (auxiliary axis)		81	Overview of enabled elements	
15	Actual a-value display (auxiliary axis)		88	Utilization display for a axis (auxiliary)	

Code numbers for “symbols”			Code numbers for “symbols”		
16	Actual b-value display (auxiliary axis)		89	Utilization display for a axis (auxiliary)	
17	Actual c-value display (auxiliary axis)		90	Utilization display for a axis (auxiliary)	
21	Tool display with compensation values (DX, DZ)		91	Utilization display for spindle	
22	Tool display with identification number		92	Utilization display for X axis	
23	Additive compensation		93	Utilization display for Z axis	
25	Tool display with tool life information		94	Utilization display for C axis	
26	Display for multipoint tools with compensation values (DX, DZ)		95	Load display Y axis	
30	Distance to go in U and actual position (auxiliary axis)		96	Utilization display for U axis (auxiliary)	
31	Distance to go in V and actual position (auxiliary axis)		97	Utilization display for V axis (auxiliary)	
32	Distance to go in W and actual position (auxiliary axis)		98	Utilization display for W axis (auxiliary)	
33	Distance to go in a and actual position (auxiliary axis)		99	Empty field	

7.5 Set-Up Parameters



Recommendation: Use “Cur. Para > Set-up (menu) – ... ” to edit the parameters. In the other menu items the parameters are listed without the axes.

Set-up parameters

Workpiece datum

- **Datum position, main spindle X, Y, Z - Slide 1**
- **Datum position, main spindle X, Y, Z - Slide 2**
- . . .
- **Datum position, main spindle X, Y, Z - Slide 1**
- **Datum position, main spindle X, Y, Z - Slide 2**
- . . .

The CNC PILOT manages the following zero points for each slide:

- Workpiece zero point, main spindle (reference: machine zero point)
- Workpiece zero point, opposing spindle (reference: machine zero point, opposing spindle) The “Workpiece zero point, opposing spindle” is derived from “Machine zero point - Zero offset” (MP 1114, 1164, ..) It is activated with “G30 H1 ...”

Notes:

- Set the workpiece zero point in the Manual Control mode.
- “PgUp/PgDn” switches to the next/previous slides.

Tool change position

- **Datum position, main spindle X, Y, Z - Slide 1**
- **Datum position, main spindle X, Y, Z - Slide 2**
- . . .

The CNC PILOT manages the tool change point for each slide (reference: machine zero point).

Notes:

- Set the tool change position in the Manual Control mode.
- “PgUp/PgDn” switches to the next/previous slides.

Zero point - Oversize G53/G54/G55

- **Oversize X, Y, Z - Slide 1**
- **Oversize X, Y, Z - Slide 2**
- . . .

The CNC PILOT manages allowances for zero point offsets for each slide.

“PgUp/PgDn” switches to the next/previous slides.

Zero point shift, C axis

- **Zero point shift, C axis 1**
- **Zero point shift, C axis 2**

Note: The zero point shift G152 is added to this parameter.

Set-up parameters

Tool life monitoring

- **Tool life switch** (tool life monitoring/quantity monitoring)
  - 0: Off
  - 1: On
- **Load monitoring**
  - 0: Off
  - 1: On

Additive compensation

- **Compensation 901..916 in X**
- **Compensation 901..916 in Z**

The CNC PILOT manages 16 compensation values (X and Z) that can be activated/deactivated in the NC program (see G149, G149 Geo).

If an additive compensation value is changed in Automatic mode, this parameter is changed accordingly.

Deletion Level/Cycle

- **Deletion level [0..9]**
- **Deletion cycle [0..99]**
  - 0: NC blocks with this deletion level are never executed.
  - 1: NC blocks with this deletion level are always executed.
  - 2..99: NC blocks with this deletion level are executed every Nth time.

A skip level can be assigned a skip cycle. As a result, NC blocks containing the specified skip level are executed each nth time.

Activate/deactivate the skip levels in automatic mode.

7.6 Machining Parameters

Machining parameters are used by the work plan generation (TURN PLUS) and various machining cycles.

1 – Global parameters for finished parts

Global parameters for finished parts

Peak-to-valley type [ORA]

Type of peak-to-valley data

- 0: No peak-to-valley data
- 1 – Rt: Peak-to-valley in [µm]
- 2 – Ra: Mean peak-to-valley in [µm]
- 3 – Rz: Mean peak-to-valley height in [µm]
- 4 – Vr: Direct feed rate value in [mm/rev]

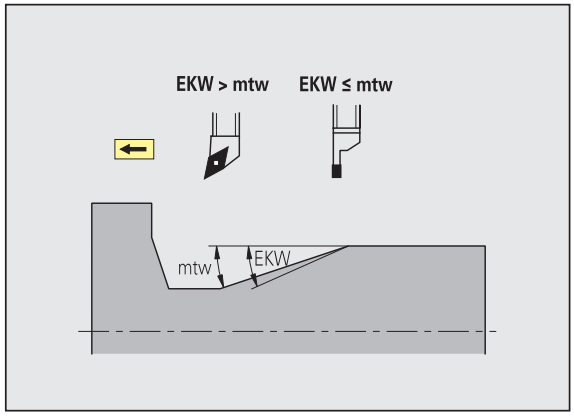
Peak-to-valley [ORW]

Peak-to-valley or feed-rate values

Permissible inward copying angle [EKW]

Tolerance angle for recess areas, used for distinguishing turning from recessing (mtw = contour angle).

- EKW > mtw: Relief turn
- EKW <= mtw: Undefined recess (no form element)



ORA, **ORW** are evaluated in the finishing cycle G890.

2 – Global technology parameters

Global technology parameters – Tools

Tool selection, tool change, speed limitation

Tool off .. [WD]

When selecting the tool, TURN PLUS considers the following data:

- 1: Current turret assignment
- 2: Primarily the current turret assignment and, additionally, the tool database.
- 3: Tool database

TURN PLUS turret [RNR]

Specify which turret assignment is to be used (requires "WD=1 or WD=2"):

- 0: Turret assignment currently selected in Machine mode
- 1: TURN PLUS turret assignment (see "Setting up and managing tool lists" on page 490)

Traversing mode to tool change position [WP]

WP determines the mode of approach and the tool change position. The sequence in which the axes are traversed is defined in the IWG or, for the AWG, in the respective machining parameters.

- 1 – IWG: The tool change position is approached at rapid traverse (G0). The position and the strategy of traverse to the tool change position is defined in the IWG.
- 2 – IWG: TURN PLUS generates a G14.
- 3 – IWG: Should not be used.
- 1 – AWG: The tool change position is approached at rapid traverse G0.
- 2 – AWG: The tool change position is approached with G14.
- 3 – AWG: TURN PLUS calculates the optimum tool change position from the current and the following tool. It then approaches this position with G0.

Speed limiting [SMAX]

Global speed limiting. You can define a small speed limit in the program head of the TURN PLUS program (see "Program Head" on page 393).

Global safety clearance

- **External on blank part [SAR]**
- **Internal on blank part [SIR]**

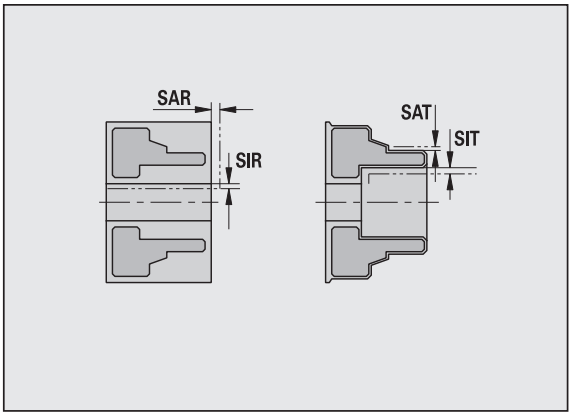
  TURN PLUS takes account of **SAR/SIR**

  - for all rough-turning operations
  - for centric predrilling

- **External on machined part [SAT]**
- **Internal on machined part [SIT]**

  TURN PLUS takes **SAT/SIT** into account on premachined workpieces for:

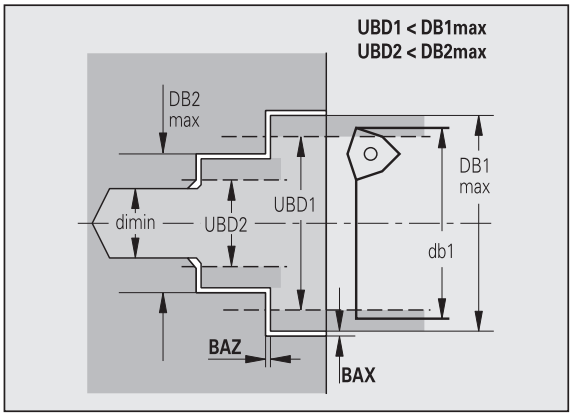
  - Finish-machining
  - Recess turning
  - Contour recessing (contour cutting)
  - Recessing
  - Thread cutting
  - Measuring



3 – Centric predrilling

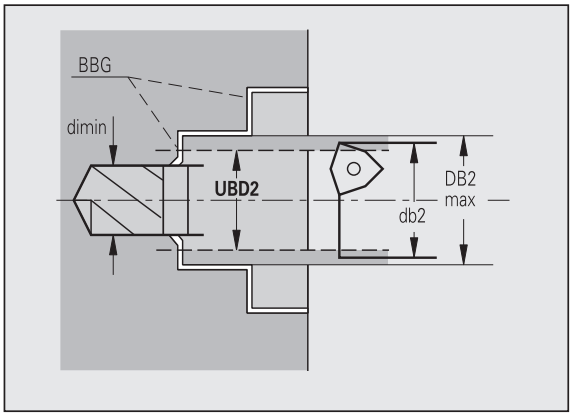
Centric predrilling – Tool selection

Tool selection
1. 1st drilling diameter limit [UBD1]
■ 1. 1st predrilling step: if UBD1 < DB1max
■ Tool selection: UBD1 <= db1 <= DB1max
2nd drilling diameter limit [UBD1]
■ 2nd predrilling step: if UBD2 < DB2max
■ Tool selection: UBD2 <= db2 <= DB2max



For **predrilling**, a maximum of three drilling steps is used:

- 1st predrilling step (limit diameter UBD1)
- 2nd predrilling step (limit diameter UBD2)
- Finish-drilling step
  - Finish-drilling is performed with: dimin <= UBD2
  - Tool selection: db = dimin



Designations in the figures at right:

- db1, db2: Drill diameter
- DB1max: Maximum inside diameter for 1st drilling step
- DB2max: Maximum inside diameter for 2nd drilling step
- dimin: Minimum inside diameter
- BBG (drilling limitation elements): Contour elements intersected by UBD1/UBD2



- UBD1/UBD2 have no effect when “Centric predrilling” has been defined as main machining operation followed by “Finish-drilling” as submachining operation in the machining sequence (see “Machining sequence – Fundamentals” on page 535).
- Prerequisite: UBD1 > UBD2
- UBD2 must permit subsequent inside machining with boring bars.

Centric predrilling – Oversizes

Oversizes

Point angle tolerance [SWT]

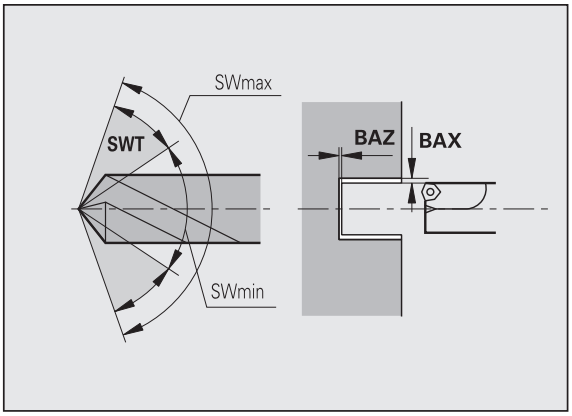
If the drilling limitation element is a diagonal element, TURN PLUS prefers using a twist drill with suitable point angle. If no suitable twist drill is available, an indexable-insert drill is selected for the predrilling operation. SWT defines the permissible point angle tolerance.

Drilling oversize – Diameter [BAX]

Machining oversize on drilling diameter (X direction – radius value).

Drilling oversize – Depth [BAZ]

Machining oversize on drilling depth (Z direction – radius value).



BAZ is not considered if:

- A subsequent inside finishing operation is not possible due to the small diameter.
- "dimin < 2* UBD2" in the finish-drilling step for blind holes.

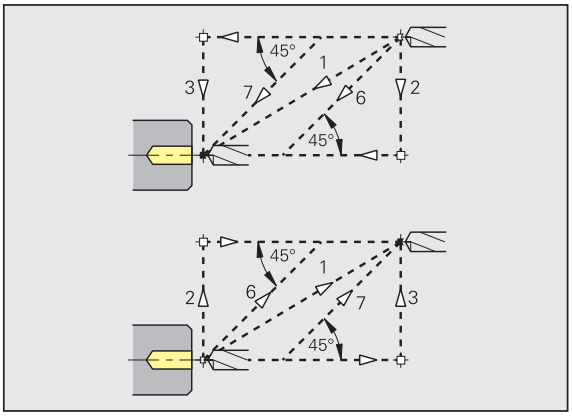
Centric predrilling – Approach and departure

Approach and departure

- **Traverse for predrilling [ANB]**
- **Traverse for tool change [ABW]**

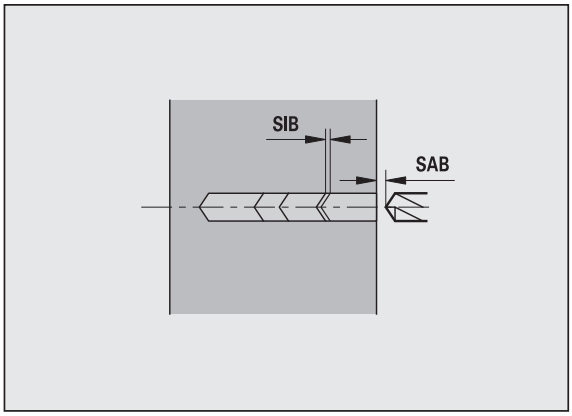
Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction



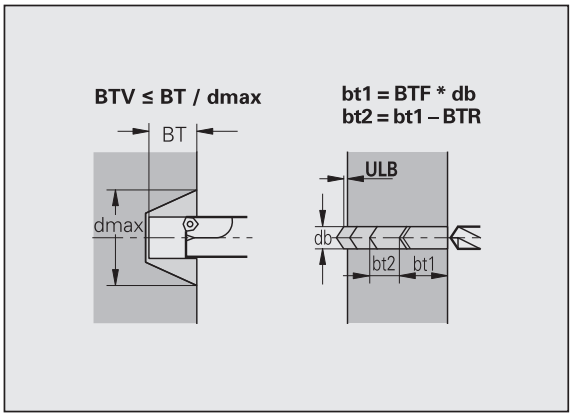
Centric predrilling – Safety clearances

Safety Clearances
Safety clearance on blank part [SAB]
Internal safety clearance [SIB] Retraction distance for deep-hole drilling (“B” for G74).



Centric predrilling – Machining

Machining
Drilling depth ratio [BTV] TURN PLUS checks the 1st and 2nd drilling steps. The predrilling step is performed with: BTV <= BT / dmax
Drilling depth factor [BTF] 1. Reduction for deep-hole drilling cycle (G74): bt1 = BTF * db
Depth reduction [BTR] Reduction for deep-hole drilling cycle (G74): bt2 = bt1 – BTR
Overhang length – Predrilling [ULB] Through-drilling length

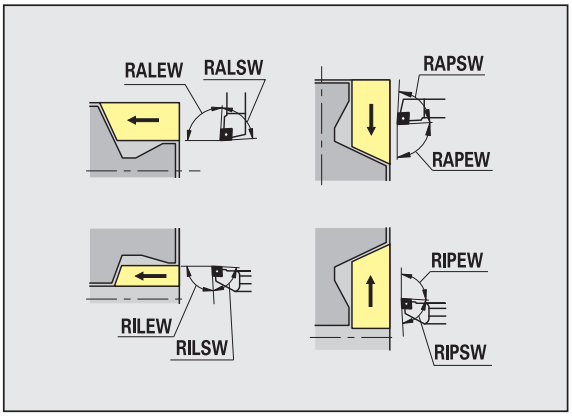


4 – Roughing

Roughing – Tool standards

Furthermore:

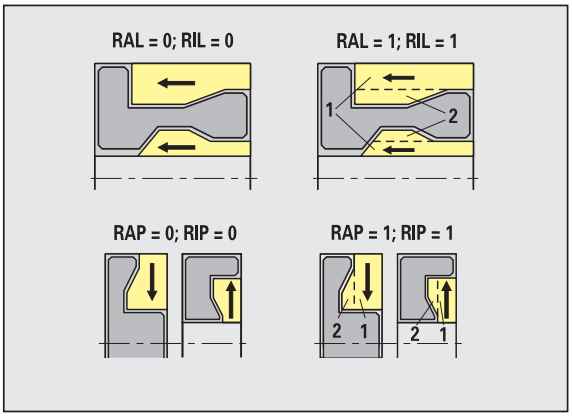
- Roughing cycles are primarily executed with standard roughing tools.
- Alternatively, tools that allow full-surface machining are used.



Tool standards

- **Tool angle – external/longitudinal [RALEW]**
- **Point angle – external/longitudinal [RALSW]**
- **Tool angle – external/transverse [RAPEW]**
- **Point angle – external/transverse [RAPSW]**
- **Tool angle – internal/longitudinal [RILEW]**
- **Point angle – internal/longitudinal [RILSW]**
- **Tool angle – internal/transverse [RIPEW]**
- **Point angle – internal/transverse [RIPSW]**

Roughing – Machining standards



Machining standards

- **Standard/Full-surface – external/longitudinal [RAL]**
- **Standard/Full-surface – internal/longitudinal [RIL]**
- **Standard/Full-surface – external/transverse [RAP]**
- **Standard/Full-surface – internal/transverse [RIP]**

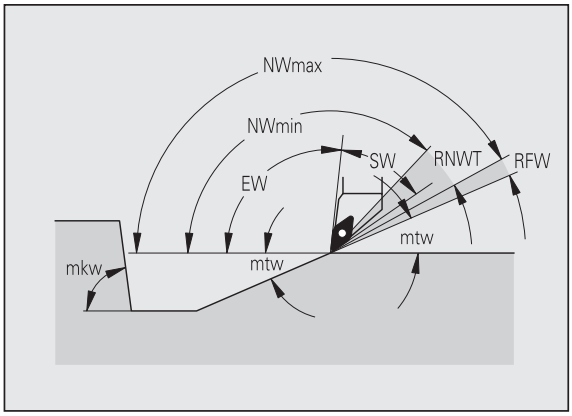
Input for RAL, RIL, RAP, RIP:

- 0: Full-surface roughing cycle, including plunge-cutting. TURN PLUS looks for a tool for full-surface machining.
- 1: Standard roughing cycle without plunge-cutting

Roughing – Tool tolerances

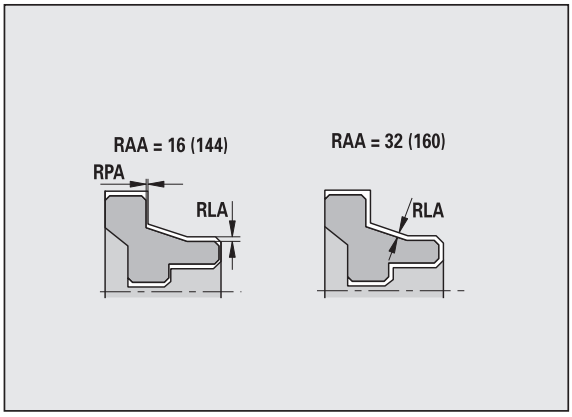
For tool selection, the following applies:

- Tool angle (EW): EW >= mkw (mkw: ascending contour angle)
- Tool angle (EW) and point angle (SW): NWmin < (EW+SW) < NWmax
- Adjacent angle (RNWT): RNWT = NWmax – NWmin



Tool tolerances
Adjacent angle tolerance [RNWT]
Tolerance range for secondary cutting edge
Relief cutting angle [RFW]
Minimum angle differential between the contour and secondary cutting edge

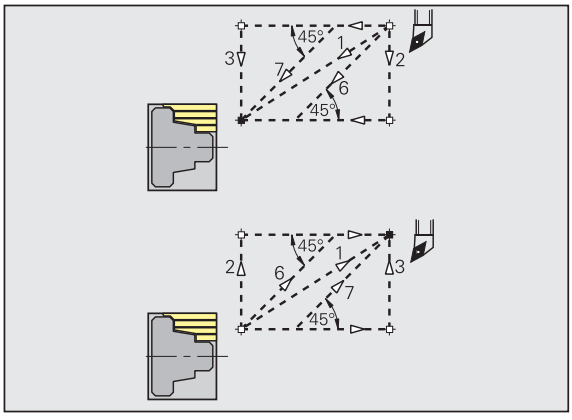
Roughing – Oversizes



Oversizes
Type of oversize [RAA]
■ 16: Longitudinal and transverse oversizes differ – no single oversizes ■ 144: Longitudinal and transverse oversizes differ – with single oversizes ■ 32: Equidistant oversize – no single oversizes ■ 160: Equidistant oversize – with single oversizes
Equidistant or longitudinal [RLA]
Equidistant oversize or longitudinal oversize
None or transverse [RPA]
Transverse oversize

Roughing – Approach and departure

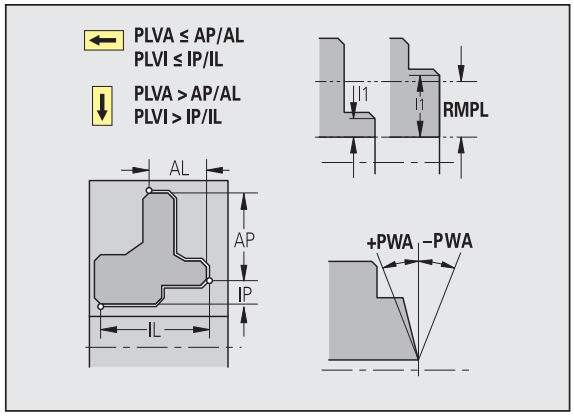
Approach and departure are in rapid traverse (G0).



Approach and departure
■ **Approach – external roughing [ANRA]** ■ **Approach – internal roughing [ANRI]** ■ **Departure – external roughing [ABRA]** ■ **Departure – internal roughing [ABRI]**
Approach/departure strategy: ■ 1: Move simultaneously in X and Z directions ■ 2: First X, then Z direction ■ 3: First Z, then X direction ■ 6: Coupled motion; X precedes Z direction ■ 7: Coupled motion; Z precedes X direction

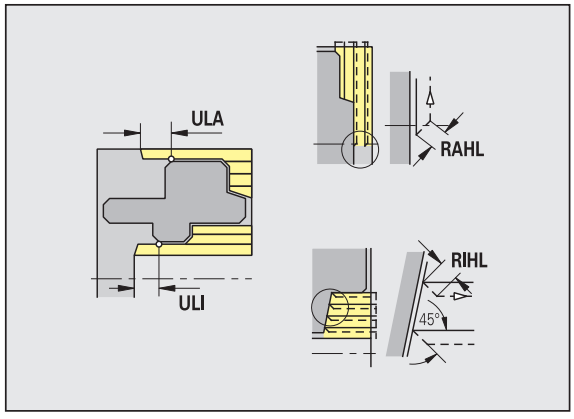
Roughing – Machining analysis

TURN PLUS uses the PLVA/PLVI parameters to define whether a roughing area is to be rough-machined longitudinally or transversely.



Machining analysis
Transverse/longitudinal ratio – external [PLVA]
■ PLVA <= AP/AL: Longitudinal machining ■ PLVA > AP/AL: Transverse machining
Transverse/longitudinal ratio – internal [PLVI]
■ PLVI <= IP/IL: Longitudinal machining ■ PLVI >= IP/IL: Transverse machining
Minimum roughing transverse length [RMPL] (radius value)
Defines whether transverse roughing is used for the front transverse element of the outside contour of a finish part. ■ RMPL > I1: Without additional transverse roughing ■ RMPL < I1: With additional transverse roughing ■ RMPL = 0: Special case
Transverse angle variation [PWA]
The first front element is declared a transverse element when it is within +PWA and –PWA.

Roughing – Machining cycles



Fixed cycles
Overhang length outside [ULA]
Relative length for external rough-machining enabling roughing beyond the target position in longitudinal direction. ULA is not considered when the cutting limitation is in front of or within the overhang.
Overhang length inside [ULI]
■ Relative length for internal rough-machining enabling roughing beyond the target position in longitudinal direction. ULA is not considered when the cutting limitation is in front of or within the overhang. ■ This parameter is used for calculating the drilling depth for centric predrilling. (see “Machining information” on page 558)
Retracting length outside [RAHL]
Retracting length for smoothing variants (H=1, 2) of roughing cycles G810 and G820 for external machining (RAHL).
Retracting length inside [RIHL]
Retracting length for smoothing variants (H=1, 2) of roughing cycles G810 and G820 for internal machining (RIHL).

Fixed cycles

Cutting depth reduction factor [SRF]

For rough-machining with tools machining opposite to the main machining direction, the infeed value (cutting depth) is reduced.

Infeed (P) for roughing cycles (G810, G820):

P = ZT * SRF

(ZT: Infeed value from technology database)

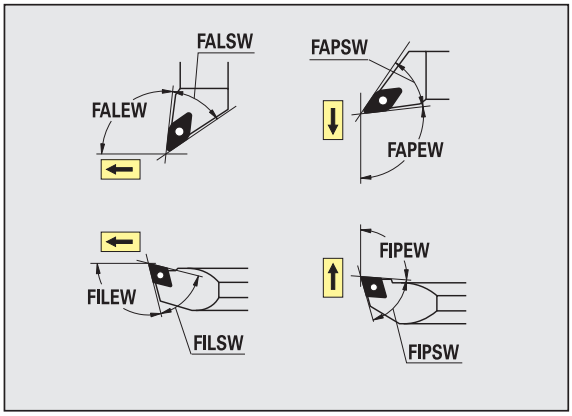
5 – Finishing

Finishing – Tool standards

TURN PLUS uses the tool angle and point angle to select the tools according to machining location and main machining direction (MMD).

Furthermore:

- Finishing cycles are primarily executed with standard finishing tools.
- If form elements such as recesses (type FD) and undercuts (type E, F, G) cannot be machined with a standard finishing tool, one form element after the other is skipped. TURN PLUS starts a renewed attempt to machine the remaining contour with the standard tool. Subsequently, the skipped form elements are machined individually with a suitable finishing tool.



Tool standards

- **Tool angle – external/longitudinal [FALEW]**
- **Point angle – external/longitudinal [FALSW]**
- **Tool angle – external/transverse [FAPEW]**
- **Point angle – external/transverse [FAPSW]**
- **Tool angle – internal/longitudinal [FILEW]**
- **Point angle – internal/longitudinal [FILSW]**
- **Tool angle – internal/transverse [FIPEW]**
- **Point angle – internal/transverse [FIPSW]**

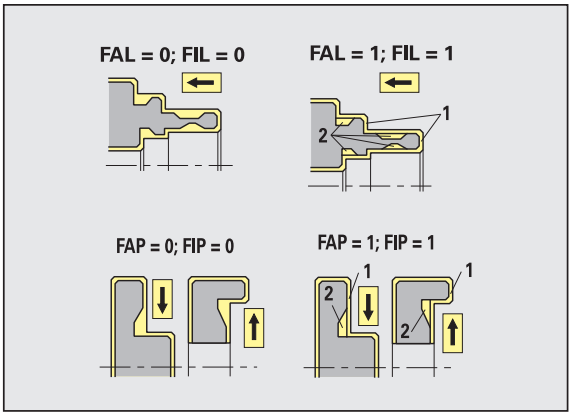
Finishing – Machining standards

Machining standards

- **Standard/Full-surface – external/longitudinal [FAL]**
- **Standard/Full-surface – internal/longitudinal [FIL]**
- **Standard/Full-surface – external/transverse [FAP]**
- **Standard/Full-surface – internal/transverse [FIP]**

Machining of contour areas:

- 0 – Full-surface finishing cycle: TURN PLUS looks for an optimum tool for machining the complete contour area.
- 1 – Standard finishing cycle:
  - The machining operation is primarily executed with standard finishing tools. Relief turns and undercuts are machined with a suitable tool.
  - If the standard finishing tool cannot be used for relief turns and undercuts, TURN PLUS splits up the machining operation into standard machining operations and machining the form elements.
  - If a division into standard machining and form-element machining is not possible, TURN PLUS switches to "full-surface machining".



Finishing – Tool tolerances

For tool selection, the following applies:

- Tool angle (EW): EW >= mkw
  (mkw: ascending contour angle)
- Tool angle (EW) and point angle (SW):
  NWmin < (EW+SW) < NWmax
- Adjacent angle (FNWT): FNWT = NWmax – NWmin

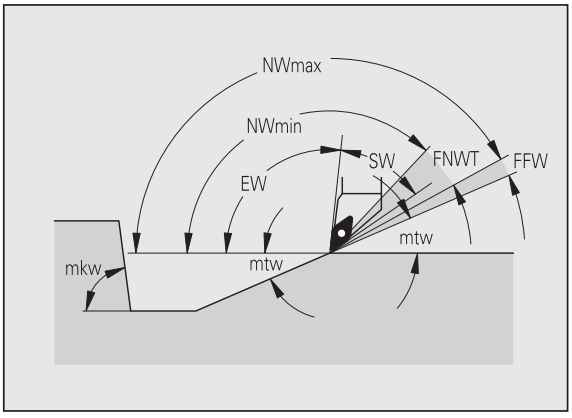
Tool tolerances

Adjacent angle tolerance [FNWT]

Tolerance range for secondary cutting edge

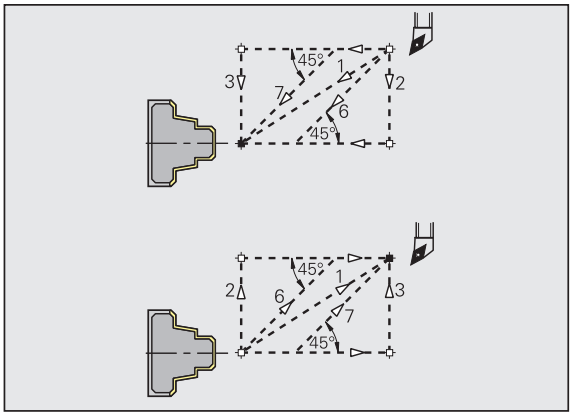
Relief cutting angle [FFW]

Minimum angle differential between the contour and secondary cutting edge



Finishing – Tool tolerances

Approach and departure are in rapid traverse (G0).



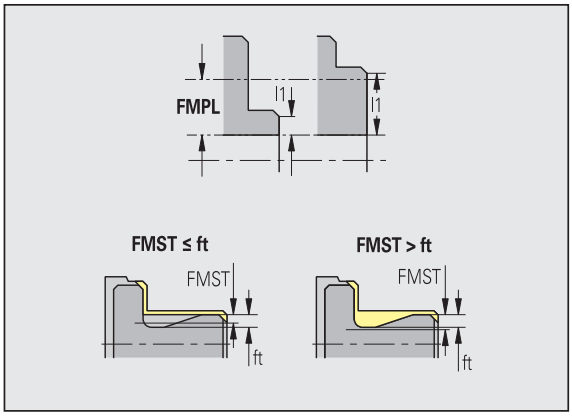
Approach and departure

- **Approach – external finishing [ANFA]**
- **Approach – internal finishing [ANFI]**
- **Departure – external finishing [ABFA]**
- **Departure – internal finishing [ABFI]**

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Finishing – Machining analysis



Machining analysis

Minimum finishing transverse length [FMPL]

TURN PLUS checks the frontmost element of the outside contour to be finish-machined. The following rules apply:

- Without inside contour: Always with additional transverse cut
- With inside contour – FMPL >= l1: Without additional transverse cut
- With inside contour – FMPL < l1: With additional transverse cut

Maximum finishing cut depth [FMST]

FMST defines the permissible infeed depth for non-machined undercuts. The finishing cycle (G890) uses this parameter to determine whether undercuts (type E, F, G) will be machined with a contour-finishing operation. The following rules apply:

- FMST > ft: With undercut machining (ft: undercut depth)
- FMST <= ft: Without undercut machining

Number of revolutions for chamfer or rounding [FMUR]

The feed rate is reduced such that at least FMUR revolutions can be executed (evaluation: finishing cycle G890).

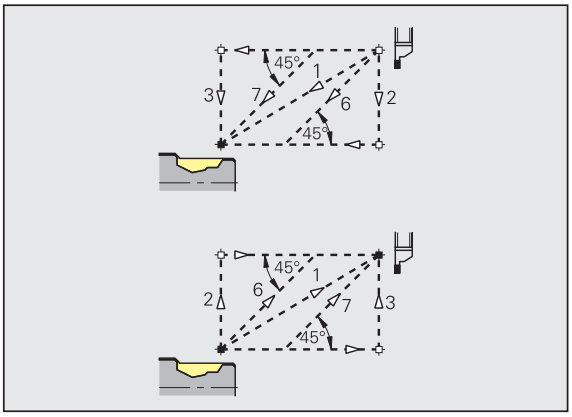
The following rules apply to FMPL:

- The additional transverse cut is executed from the outside toward the inside.
- The "Transverse angle variation PWA" has no effect on the analysis of the transverse elements.

6 – Recessing and contour recessing

Recessing, contour recessing – Approach and departure

Approach and departure are in rapid traverse (G0).



Approach and departure

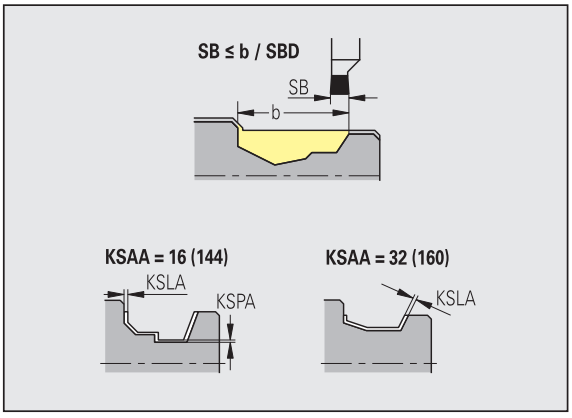
- **Approach – external recessing [ANESA]**
- **Approach – internal recessing [ANESI]**
- **Departure – external recessing [ABESA]**
- **Departure – internal recessing [ABESI]**

- **Approach – external contour recessing [ANKSA]**
- **Approach – internal contour recessing [ANKSI]**
- **Departure – external contour recessing [ABKSA]**
- **Departure – internal contour recessing [ABKSI]**

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Recessing, contour recessing – Tool selection, oversizes



Tool selection, oversizes

Recessing width divisor [SBD]

If a recess base machined with a contour-recessing cycle does not contain paraxial elements but only linear elements, a suitable tool is selected using the "recessing width divisor SBD."

SB <= b / SBD

(SB: width of recessing tool; b: width of machining area)

Type of oversize [KSAA]

The recessing area to be machined can be assigned oversizes. When oversizes have been defined, a recess is first rough-machined and then finish-machined. Inputs:

- 16: Longitudinal and transverse oversizes differ – no single oversizes
- 144: Longitudinal and transverse oversizes differ – with single oversizes
- 32: Equidistant oversize – no single oversizes
- 160: Equidistant oversize – with single oversizes

Tool selection, oversizes

Equidistant or longitudinal [KSLA]

Equidistant oversize or longitudinal oversize

None or transverse [KSPA]

Transverse oversize

- The oversizes are accounted for when machining contour valleys with a contour-recessing operation.
- Standardized recesses such as recess types D, S, A are completed in one machining cycle. A division into rough-machining and finish-machining is only possible in DIN PLUS.

Recessing, contour recessing – Machining

Evaluation: DIN PLUS

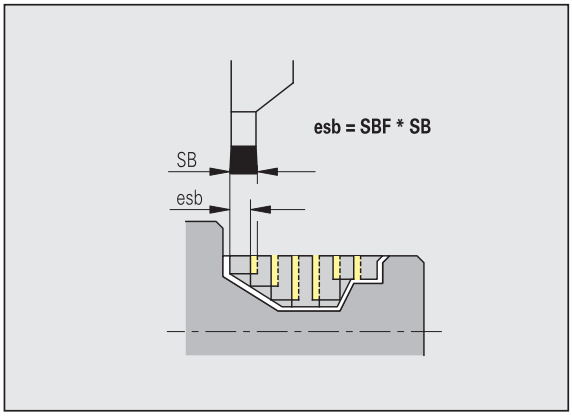
Machining

Recessing width factor [SBF]

SBF is used for calculating the maximum offset for recessing cycles G860, G866:

esb = SBF * SB

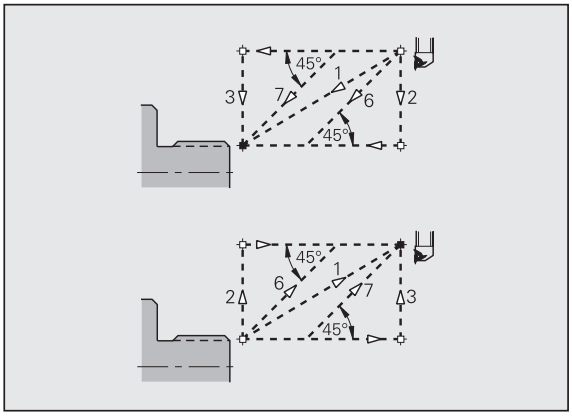
(esb: effective recessing width; SB: width of recessing tool)



7 – Thread cutting

Thread cutting – Approach and departure

Approach and departure are in rapid traverse (G0).



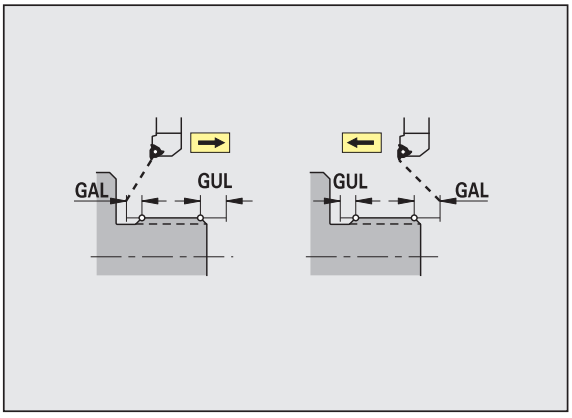
Approach and departure

- **Approach – external thread [ANGA]**
- **Approach – internal thread [ANGI]**
- **Departure – external thread [ABGA]**
- **Departure – internal thread [ABGI]**

  Approach/departure strategy:

  - 1: Move simultaneously in X and Z directions
  - 2: First X, then Z direction
  - 3: First Z, then X direction
  - 6: Coupled motion; X precedes Z direction
  - 7: Coupled motion; Z precedes X direction

Thread cutting – Machining



Machining

Thread starting length [GAL]

Starting length before threading cut.

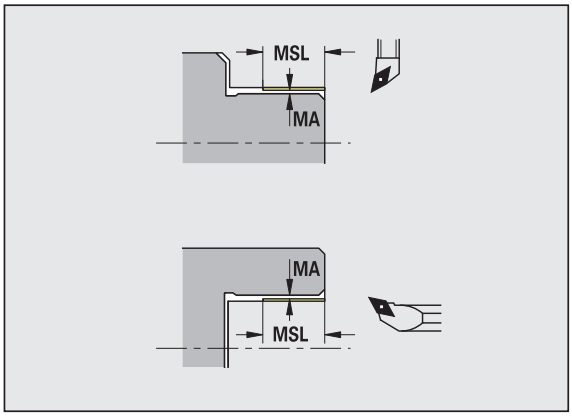
Thread run-out length [GUL]

Overrun length after threading cut.

GAL/GUL are automatically transferred to the thread attributes “starting length B/overrun length P” if they have not been entered as attributes.

8 – Measuring

The measuring parameters are assigned to the fit elements as an attribute.



Measurement procedure

Measurement type [MART]

1: Manual measurement – calls expert program

Measuring loop counter [MC]

Defines the measurement/loop intervals.

Current consumption [mA]

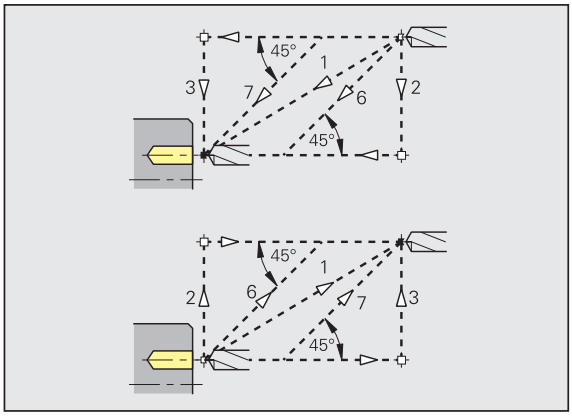
Oversize still applied to the element to be measured.

Measuring cut length [MSL]

9 – Drilling

Drilling – Approach and departure

Approach and departure are in rapid traverse (G0).



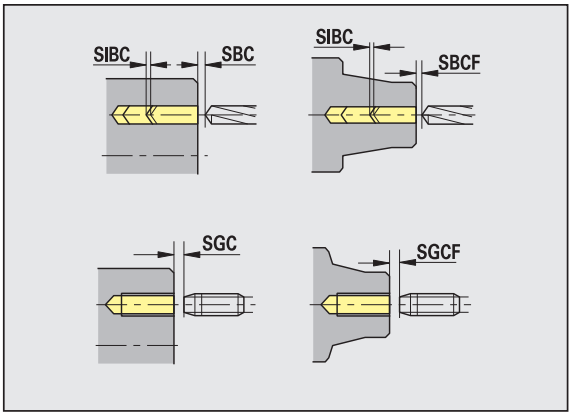
Approach and departure

- **Approach – end face [ANBS]**
- **Approach – lateral surface [ANBM]**
- **Departure – end face [ABGA]**
- **Departure – lateral surface [ABGI]**

  Approach/departure strategy:

  - 1: Move simultaneously in X and Z directions
  - 2: First X, then Z direction
  - 3: First Z, then X direction
  - 6: Coupled motion; X precedes Z direction
  - 7: Coupled motion; Z precedes X direction

Drilling – Safety clearances



Safety Clearances

Internal safety clearance [SIB]

Retraction distance for deep-hole drilling (“B” for G74).

Driven drills [SBC]

Safety clearance for driven tools on end face and lateral surface.

Stationary drills [SBCF]

Safety clearance on end face and lateral surface for tools that are not driven.

Driven taps [SGC]

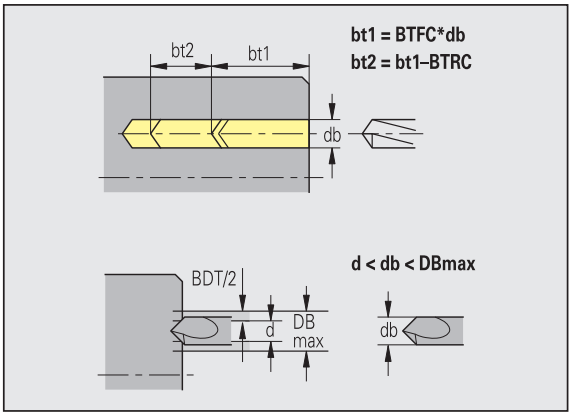
Safety clearance for driven tools on end face and lateral surface.

Stationary taps [SGCF]

Safety clearance on end face and lateral surface for tools that are not driven.

Drilling – Machining

The parameters apply to drilling with deep-hole drilling cycle (G74).



Machining

Drilling depth factor [BTFC]

1st drilling depth: bt1 = BTFC * db

(db: drill diameter)

Depth reduction [BTRC]

2nd drilling depth: bt2 = bt1 – BTRC

The subsequent drilling steps are reduced accordingly.

Diameter tolerance for drill [BDT]

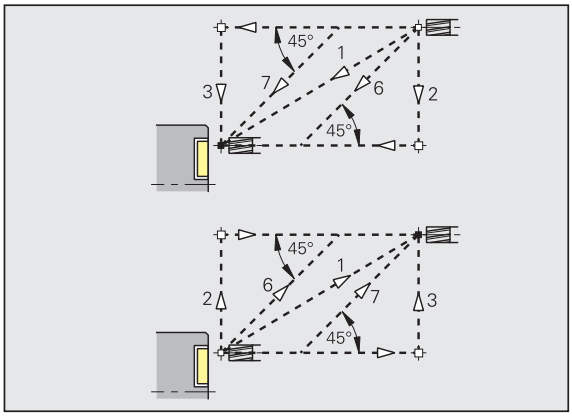
For selecting the desired drill (centering drills, countersinks, stepped drill, taper reamers).

- Drilling diameter: DBmax = BDT + d (DBmax: maximum drilling diameter)
- Tool selection: DBmax > DB > d

10 – Milling

Milling – Approach and departure

Approach and departure are in rapid traverse (G0).



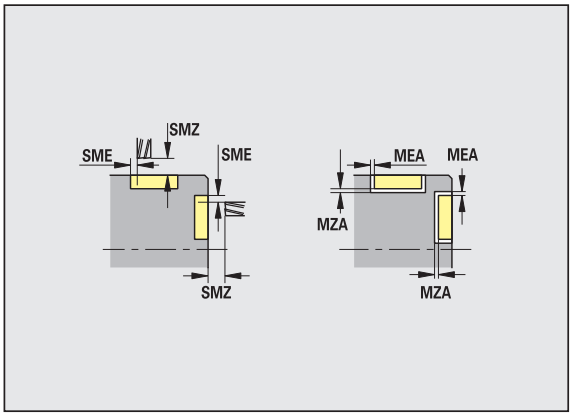
Approach and departure

- **Approach – end face [ANMS]**
- **Approach – lateral surface [ANMM]**
- **Departure – end face [ABMA]**
- **Departure – lateral surface [ABMM]**

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Milling – Safety clearances and oversizes



Safety clearances and oversizes

Safety clearance in infeed direction [SMZ]

Distance between starting position and top edge of object to be milled.

Safety clearance in milling direction [SME]

Distance between milling contour and side of mill.

Oversize in milling direction [MEA]

Oversize in infeed direction [MZA]

Load monitoring

11 – Load monitoring – General switches

Load monitoring – General switches
Load monitoring ON/OFF ■ 0 – OFF: TURN PLUS does not generate any commands for load monitoring ■ 1 – ON: TURN PLUS generates commands for load monitoring
Component positions Corresponds to the parameter Q of G996: ■ 0: Monitoring not active ■ 1: Do not monitor rapid traverse ■ 2: Monitor rapid traverse

12..19 – Load monitoring for machining modes

The first parameter switches load monitoring on and off. The other parameters define the components to be monitored with regard to machining location/machining mode.

Load monitoring for machining modes	
Inputs: "Machining mode ..." ON/OFF: ■ 0: Load monitoring OFF ■ 1: Load monitoring ON Components to be monitored (for more than one component, sum of the component codes): ■ 0: No monitoring ■ 1: X axis ■ 2: Y axis	Components to be monitored (continued): ■ 4: Z axis ■ 8: Main spindle ■ 16: Driven tool ■ 32: Spindle 3 ■ 64: Spindle 4 ■ 128: C axis 1
12 Load monitoring for centric predrilling ■ Centric drilling ON/OFF ■ Centering ■ Drilling ■ Boring ■ Countersinking ■ Reaming ■ Tapping	**16 Load monitoring for recessing** ■ Recessing ON/OFF ■ External ■ Internal
13 Load monitoring for roughing ■ Roughing ON/OFF ■ External/longitudinal ■ External/transverse ■ Internal/longitudinal ■ Internal/transverse	**17 Load monitoring for thread cutting** ■ Thread cutting ON/OFF ■ External ■ Internal ■ Transverse

Load monitoring for machining modes	
14 Load monitoring for contour recessing ■ Precutting ON/OFF ■ External ■ Internal ■ Transverse	**18 Load monitoring for drilling in C axis** ■ Drilling in C axis ON/OFF ■ Centering ■ Drilling ■ Boring ■ Countersinking ■ Reaming ■ Tapping
15 Load monitoring for contour machining ■ Finish-machining ON/OFF ■ External ■ Internal	**19 Load monitoring for milling in C axis** ■ Milling ON/OFF ■ Key-way milling ■ Contour milling ■ Pocket milling ■ Deburring ■ Engraving

20 – Direction of rotation for rear-side machining

Rear-side machining
Mirroring the direction of rotation ■ 0: Same direction of rotation for front-side and rear-side machining ■ 1: Mirror the direction of rotation (M4 instead of M3; M3 instead of M4)

21 – Name of the subroutines

TURN PLUS uses expert programs for functions such as workpiece transfer for full-surface machining. In this parameter you can define which expert programs (subprograms) are to be used. Enter the subprogram names.

Expert programs
SP 100098: Parting
SP 100099: Bar loader
SP EXUMS12 (currently no meaning)
SP EXUMS12A (currently no meaning)
SP MEAS01: Measuring cut
SP UMKOMPL: Rechucking for lathes with opposing spindle
SP UMKOMPLA: Parting and rechucking for lathes with opposing spindle
SP UMHAND: Rechucking for lathes without opposing spindle
SP ABHAND: Parting and rechucking for lathes without opposing spindle

22 – Sequence of tool selection

If more than one slide is used for machining, define the sequence in which TURN PLUS is to load the tool carriers. Enter the slide numbers one after the other, without separators (example: "351" means $3, then $5, then $1).

Sequence of tool selection
1st setup [123456] Sequence in which TURN PLUS loads the tool carriers for the first setup.
2nd setup [123456] Sequence in which TURN PLUS loads the tool carriers for the second setup.

23 – Template management

Available as of software version 625 952-05.

Select whether constants are to be output when templates are used.

Template management

Output of constants with templates

- 0: Without output of constants
- 1: With output of constants

24 – Parameter of the rechucking subroutines

Available as of software version 625 952-05.

With this parameter you influence the transfer parameters of the rechucking subroutines. The following entries do **not** influence the standard subroutines UMKOMPL and UMKOMPLA (see machining parameter 21).

Parameters of the rechucking subroutines

EXPERT - LA

- –99999: parameter transfer
- –99998: no parameter transfer
- Other numerical values: the entered numerical value is transferred

EXPERT - LB

- –99999: parameter transfer
- –99998: no parameter transfer
- Other numerical values: the entered numerical value is transferred

. . .



8

Operating Resources

8.1 Tool Database

The CNC PILOT stores up to 999 tool descriptions which are managed via the tool editor.

Data exchange and data backup: The CNC PILOT supports data exchange and data backup of operating resources (tools, chucking equipment, technology data) and the associated fixed-word list (see “Parameters and Operating Resources” on page 674).



Tools that do not fit into any of the standard tool type groups are assigned to special turning/drilling/milling tools. They are not used for contour cycles and are not used by TURN PLUS..

Tool editor

Editing the tool data

The tool data are edited in three dialog boxes. The parameters of the first two dialog boxes depend on the tool type. The third dialog box serves for multitool management and tool life management. Edit the third dialog box "as required."

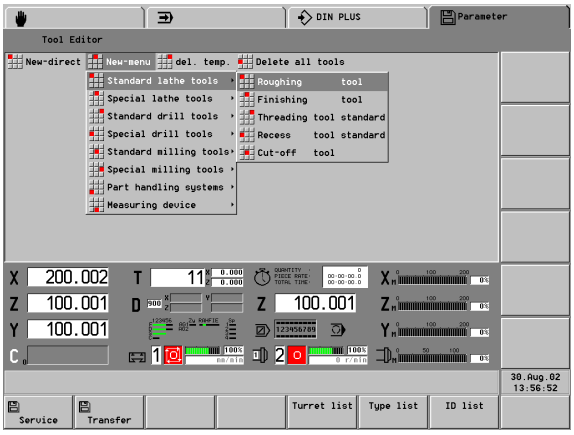
The tool parameters include:

- Fundamental data
- Information on tool depiction (simulation/control graphics)
- Information for TURN PLUS (tool selection, automatic working plan generation)

The data can be omitted provided that you do not use TURN PLUS, or do not wish tools to be depicted.

Call the tool editor:

- Select “Tool” in the Parameter mode.



Soft keys	
Service	Change to Service mode
Transfer	Change to Transfer mode

Describing the new tool (entering the tool type directly)

Select “New direct.”

The tool type is known: Enter the “Tool type”.

The tool type is not known:



Press the soft key and define the tool type by entering the following data:

- Main group
- Subgroup
- Machining direction

Enter the tool data.

Describing the new tool (selecting the tool type)

Select “New menu.”

Select the tool type from the menu.

Enter the tool data.

Temporary tool descriptions: Tools that are not permanently stored in the tool database can be described in the NC program. These “temporary” tool descriptions start with “_SIM..” or “_AUTO..”. (see “Tool programming” on page 121).

Delete temporary tool descriptions:

- Select “del. temp.”. The tool editor deletes all temporary tools.

Tool lists

Use the tool lists as a starting point for editing, copying or deleting entries.

Abbreviations in the header of the tool list:

- rs: Cutting radius
- db: Drill diameter
- df: Cutter diameter
- ew: Tool angle
- bw: Drill angle
- fw: Cutter angle
- T no.: T number of the turret list

Calling a tool list

Turret list	The editor lists the current tool carrier assignments.
Type list	The editor lists the entries sorted by tool type.
ID list	The editor lists the entries sorted by ID number. Only those entries are displayed that correspond to the mask for ID numbers.

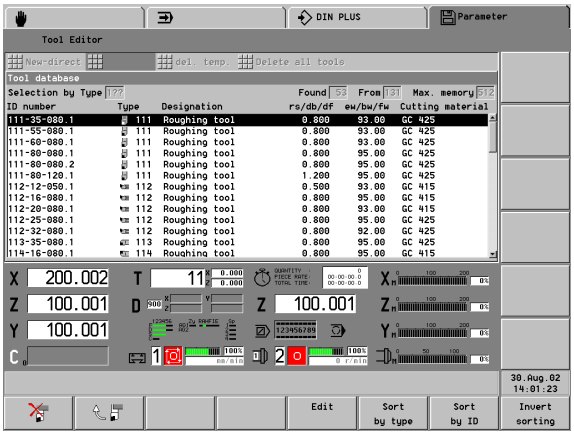
Tool type: Define the tool type by entering the following data:

- Main group
- Subgroup
- Machining direction

Mask for ID numbers:

- Enter part of the ID: Any desired (number of) characters can be specified at this position in the mask.
- "?": Any desired character can be specified at this position in the mask.

Entries of the turret list are neither copied nor deleted in the tool editor. It is possible to edit the entries when Automatic mode is not active.



Soft keys

	Delete tool entry
	Copy tool entry
Edit	Edit tool entry
Sort by type	Sort tool list by type
Sort by ID	Sort tool list by ID number
Invert sorting	Reverse the sorting sequence

Editing the tool list

Place the cursor on the tool you want to edit.



Copy the entry.



Deleting an entry



Press the soft key or Enter key. The CNC PILOT presents the tool data for editing.

Copying tools:

- You can only copy “similar” tools.
- The “new” tool is assigned a new ID number.

Displaying the tool graphic

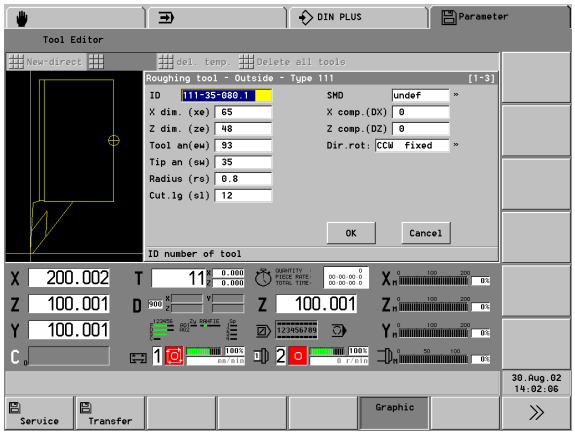
The CNC PILOT generates the displayed tool from the parameters. The graphic enables you to check the entered data. Changes become effective as soon you exit the input box.

Tool position: If the “mount type” tool parameter is used: The CNC PILOT looks for the mount type in the tool mount descriptions as of MP 511. The tool position depends on the first tool mount of the specified type.

Display the tool graphic:



▶ Press the soft key while the dialog box is open.



Exit the tool graphic:



▶ Press the soft key again.

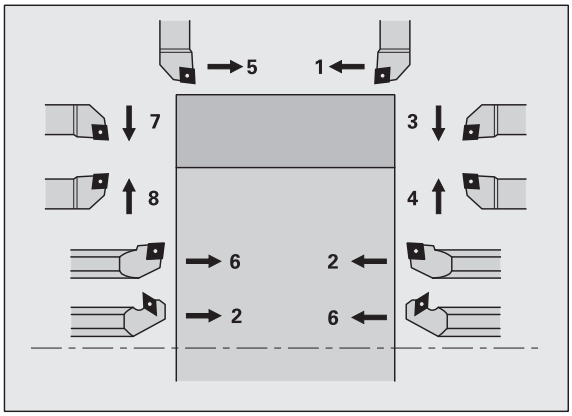
Overview of tool types

Special tools are tools that do not fit into any other type. They are not used for contour cycles and are not used by TURN PLUS..

Lathe tools

- Roughing tool (type 11x)
- Finishing tool (type 12x)
- Threading tool, standard (type 14x)
- Recessing tool (type 15x)
- Parting tool (type 161)
- Button tool (type 21x)
- Copying tool (type 22x) – TURN PLUS uses copying tools only for undercut types H and K.
- Recess-turning tool (type 26x)
- Knurling tool (type 27x)
- Special turning tool (type 28x)

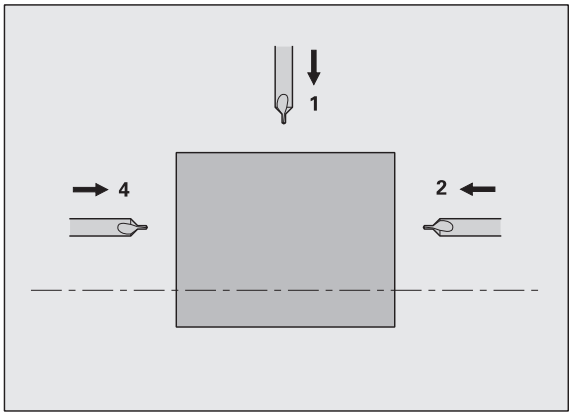
Main machining direction (third digit of tool type): see figure



Drilling tools

- Centering tool (type 31x)
- NC center drill (type 32x)
- Twist drill (type 33x)
- Indexable-insert drill (type 34x)
- Counterbore (type 35x)
- Countersink (type 36x)
- Tap (type 37x)
- Step drill (type 42x)
- Reamer (type 43x)
- Tap drill (type 44x)
- Delta drill (type 47x)
- Turn-out tool (type 48x) – not used by TURN PLUS
- Special drilling tool (type 49x)

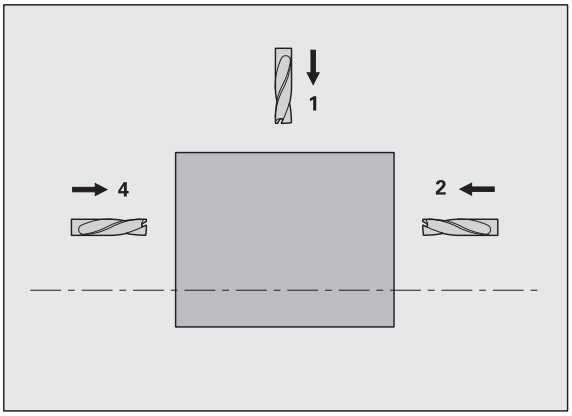
Main machining direction (third digit of tool type): see figure



Milling tools

- Twist drill cutter (type 51x)
- End milling cutter (type 52x)
- Side milling cutter (type 56x) – not used by TURN PLUS
- Angle cutter (type 61x)
- Thread milling cutter (type 63x) – not used by TURN PLUS
- Milling pins (type 64x)
- Circular saw blade (type 66x) – not used by TURN PLUS
- Special milling tool (type 67x)

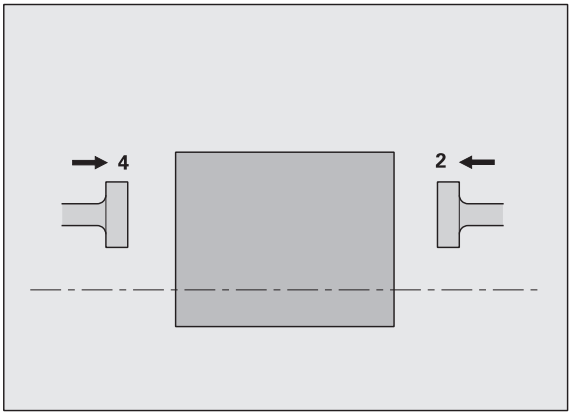
Main machining direction (third digit of tool type): see figure



Part Handling Systems

- Stopper tool (type 71x)
- Bar gripper (type 72x)
- Rotating gripping device (type 75x)

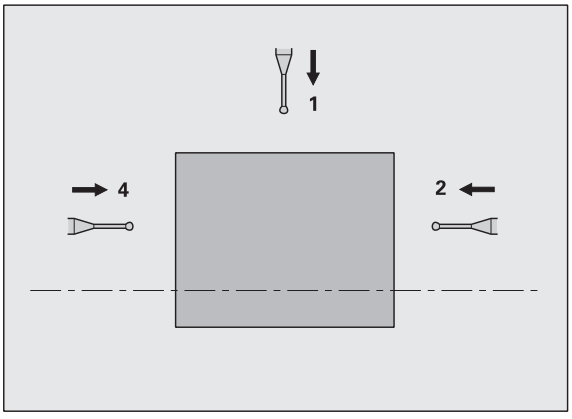
Main machining direction (third digit of tool type): see figure



Measuring systems

- Touch probe (type 81x)

Main machining direction (third digit of tool type): see figure



Tool parameters

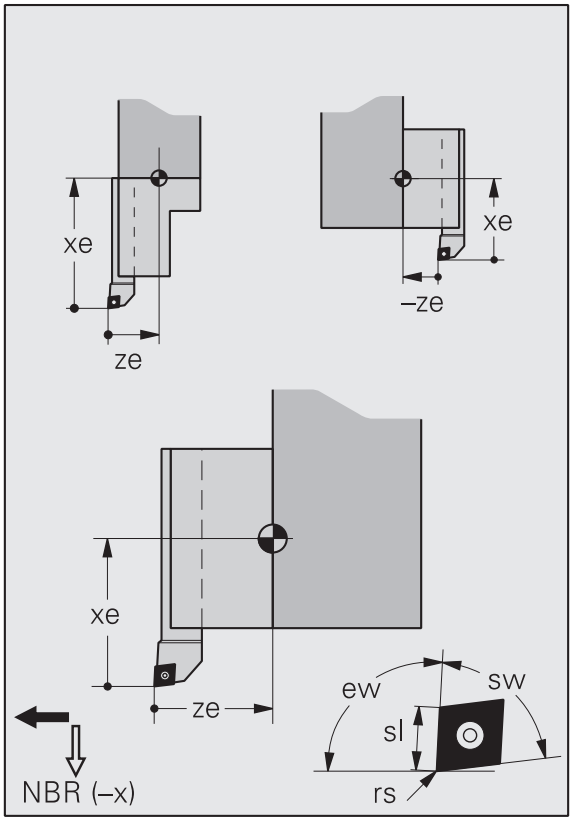
The use of the tool parameters is indicated by the following letters:

- **G:** Basic data
- **S:** Depicting the tool during simulation/control graphics
- **TP:** Information for TURN PLUS (tool selection)

Parameters for turning tools

Example: Tool type 111

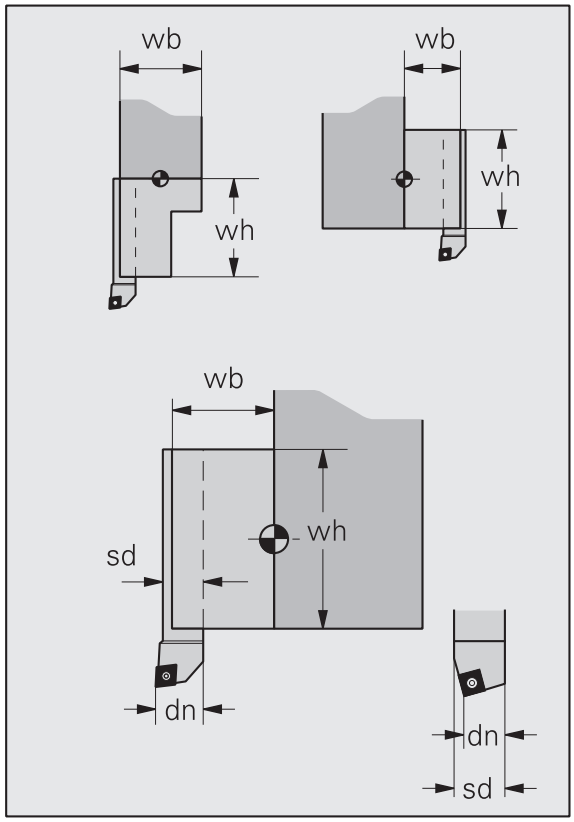
Parameters in dialog box 1	G	S	TP
ID: Tool ID number	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Tool an. (ew): Tool angle	•	•	•
Tip an. (sw): Point angle	•	•	•
Radius (rs): Cutting radius	•	•	•
Cut.wid. (sb)			
■ Recessing tool: Cutting width	•	•	•
■ Threading tool: Distance between tooth edge and tooth tip	•	•	–
■ Knurling tool: Roller width	–	•	–
Cut.lg. (sl)			
■ Knurling tool: Roller diameter	–	•	–
■ Other tools: Cutting length	•	•	•
NBR: Secondary machining direction	•	–	•
X, Z, Y comp. (DX, DZ, DY): Compensation values (maximum +/– 10 mm)	•	–	–
Dir.rot.: Direction of spindle rotation	•	–	•
Usab.lg.(nl): Usable length for internal tools	–	–	•
Dep.imm. (et): Maximum infeed depth	•	•	•
S comp. (DS): Special compensation of the third side of the tool (maximum cutting width +/– 10 mm). See also G148 and G150/G151.	•	–	–



Threading tool:

- "ze" and "xe" are measured from the tooth edge.
- The direction of rotation defines whether an overhead tool or a standard tool is used.

Parameters in dialog box 2	G	S	TP
T hold. DIN: Type of tool holder	–	•	–
T hold. heig. (wh): Height of tool holder	–	•	–
T hold. brea. (wb): Width of tool holder	–	•	–
Breadth (dn): Tool width (tool tip to shank back)	–	•	–
Shank d (sd): Shank diameter	–	•	–
Design (A) ■ Threading tool, recessing tool, parting tool, recess-turning tool: Left-hand or right-hand tool ■ Button tool with tool position 1..4: Left-hand, right-hand or neutral tool	•	•	•
Available as of software version 625 952-05. **Position angle (rw):** for cranked recessing and recess-turning tools when used with the B axis.	•	•	•
Pitch: Thread pitch	•	–	•
Available: Physical availability	–	–	•
Picture no.	–	•	–
Cutting material	–	–	•
CSP comp.: Compensation factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Depth comp.: Compensation factor for cutting depth	–	–	•
Mount type	•	–	•



- The “Design” parameter determines whether the tool reference point is located on the right/left side.
- For neutral button tools, the tool reference point is located on the left side.
- As of software version 625 952-05: With cranked recessing and recess-turning tools the recessing process must always be executed perpendicular to one of the principal axes.

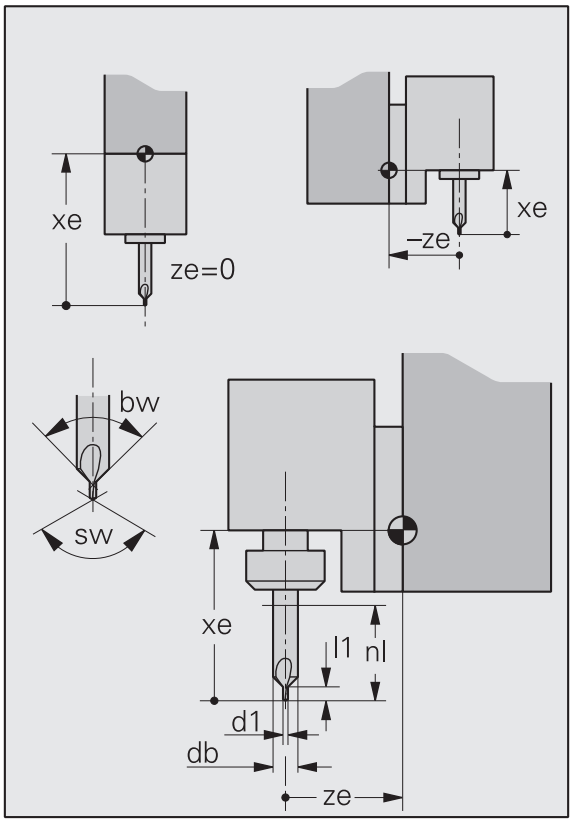
Additional information:

- Dialog box 3: see “Multipoint tools, tool life monitoring” on page 620
- see “Explanation of tool data” on page 622
- see “Tool holder, tool mount” on page 624

Drilling tool parameters

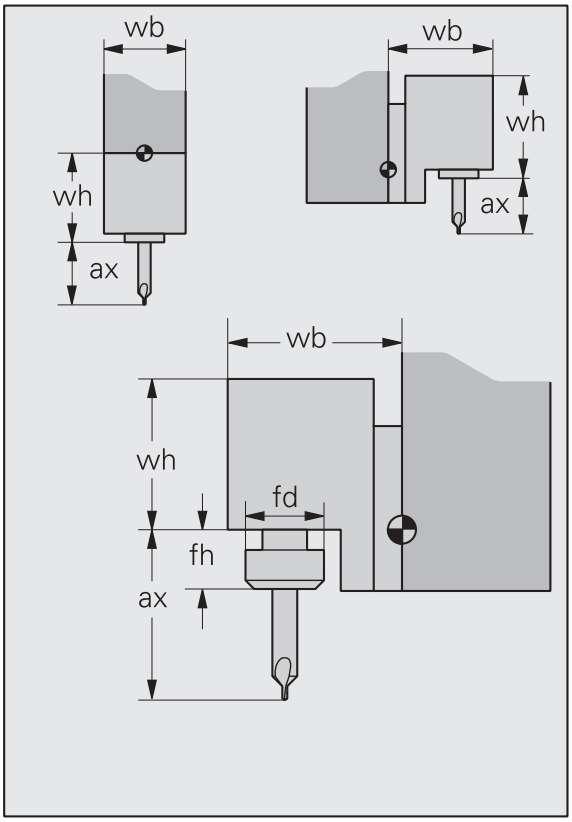
Example: Tool type 311

Parameters in dialog box 1	G	S	TP
ID: Tool ID number	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Diamet. (db): Drill diameter	•	•	•
Bor.ang. (bw): Drill angle	•	•	•
Tip an. W (sw): Point angle	•	•	•
Shnk.di. (d1): Shank diameter	•	•	•
Shnk.ln. (l1): Shank length	•	•	•
Pos.ang. (rw): Position angle	•	•	–
X, Z, Y comp. (DX, DZ, DY): Compensation values (maximum +/– 10 mm)	•	–	–
Dir.rot.: Direction of spindle rotation	•	–	•
Usab.lg.(nl): Usable length of drill	–	–	•
Type of tap:	•	–	•
■ 0: Undefined			
■ 11: Metric			
■ 12: Fine-pitch thread			
■ 13: Inch thread			
■ 14: Pipe thread			
■ 15: UNC			
■ 16: UNF			
■ 17: PG			
■ 18: NPT			
■ 19: Trapezoid thread			
■ 20: Other			
Strtlen (al): Length of first cut	•	•	•



The "Type of tap" parameter is used for determining the thread parameters; the AWG accounts for this parameter when selecting a tool.

Parameters in dialog box 2	G	S	TP
T hold. DIN: Type of tool holder	–	•	–
T hold. heig. (wh): Height of tool holder	–	•	–
T hold. brea. (wb): Width of tool holder	–	•	–
Chck.di. (fd): Diameter of chuck	–	•	–
Chck.he. (fh): Height of chuck	–	•	–
Sali.lg.(ax): Salient length	–	•	–
Pitch (hb): Thread pitch	•	–	•
Fit qual(ity): H6, H7, H8, H9, H10, H11, H12 or H13	–	–	•
Available: Physical availability	–	–	•
Picture no.	–	•	–
Cutting material	–	–	•
CSP comp.: Compensation factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Depth comp.: Compensation factor for cutting depth	–	–	•
Mount type	•	–	•



- The tool selection function of TURN PLUS only checks whether the fit quality is defined/undefined. It does not perform a detailed evaluation.
- Chuck
  - Holders F, K: “fd, fh” define the holder dimensions.
  - Other holders: Where fd=0/fh=0, no chuck is displayed.

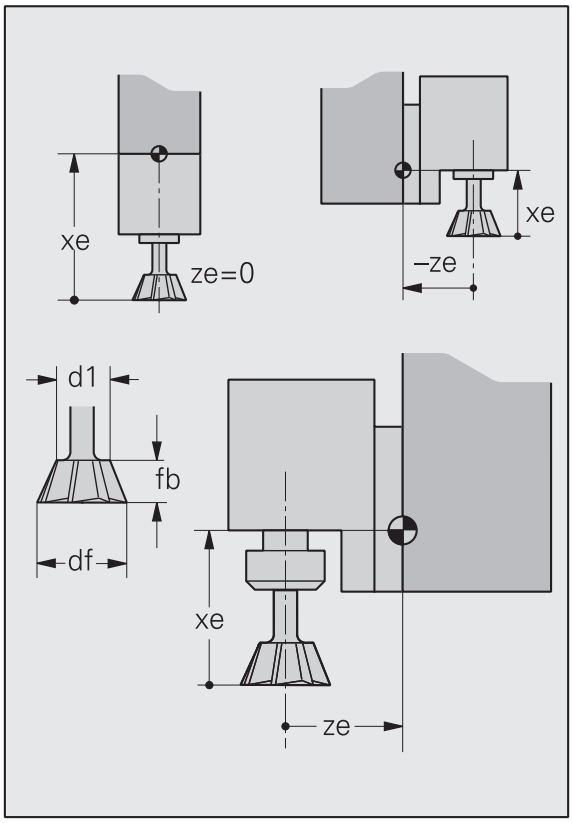
Additional information:

- Dialog box 3: see “Multipoint tools, tool life monitoring” on page 620
- see “Explanation of tool data” on page 622
- see “Tool holder, tool mount” on page 624

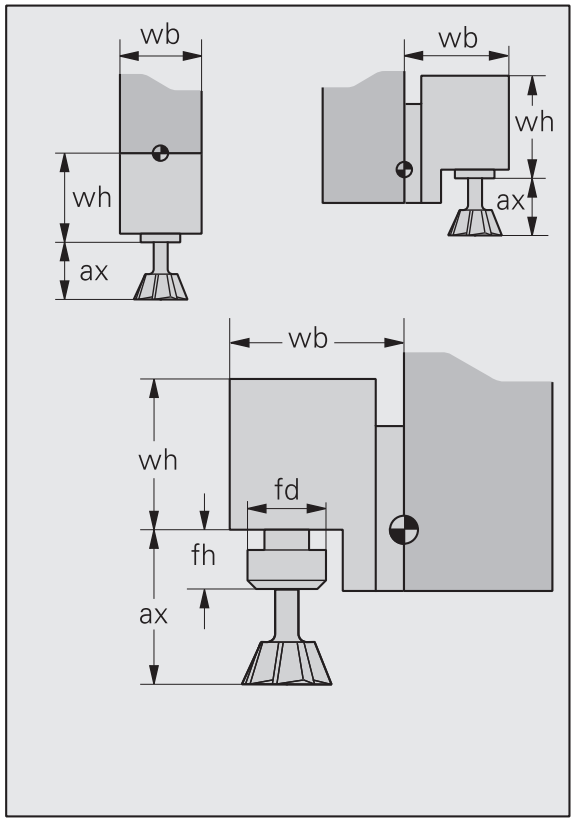
Milling cutter parameters

Example: Tool type 611

Parameters in dialog box 1	G	S	TP
ID: Tool ID number	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Diamet. (df): Cutter diameter, end	•	•	•
Diamet. (d1): Cutter diameter	•	•	•
Width (fb): Cutter width	•	•	•
Angle (fw): Cutter angle	•	•	•
Dep.imm. (et): Maximum infeed depth	•	•	–
Pos.ang. (rw): Position angle	•	•	–
X, Z, Y comp. (DX, DZ, DY): Compensation values (maximum +/– 10 mm)	•	–	–
D corr. (DD): Compensation value for cutter diameter	•	–	–
Dir.rot.: Direction of spindle rotation	•	–	•
Cut.lg.(sl): Cutting length	•	•	•
Number of teeth on the cutter	•	–	•



Parameters in dialog box 2	G	S	TP
T hold. DIN: Type of tool holder	–	•	–
T hold. heig. (wh): Height of tool holder	–	•	–
T hold. brea. (wb): Width of tool holder	–	•	–
Chck.di. (fd): Diameter of chuck	–	•	–
Chck.he. (fh): Height of chuck	–	•	–
Sali.lg.(ax): Salient length	–	•	–
Pitch (hf): Thread pitch	•	–	–
Threads per unit length (gb) for multiple threads	–	–	–
Tooth type: ■ 0: Undefined ■ 1: Front face, straight ■ 2: Front face, inclined ■ 3: Lateral surface, straight ■ 4: Lateral surface, inclined ■ 5: Front face and lateral surface, straight ■ 6: Front face and lateral surface, inclined ■ 7: Special tooth type	–	–	•
Available: Physical availability	–	–	•
Picture no.	–	•	–
Cutting material	–	–	•
CSP comp.: Compensation factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Depth comp.: Compensation factor for cutting depth	–	–	•
Mount type	•	–	•



Chuck: Where fd=0/fh=0, no chuck is displayed.

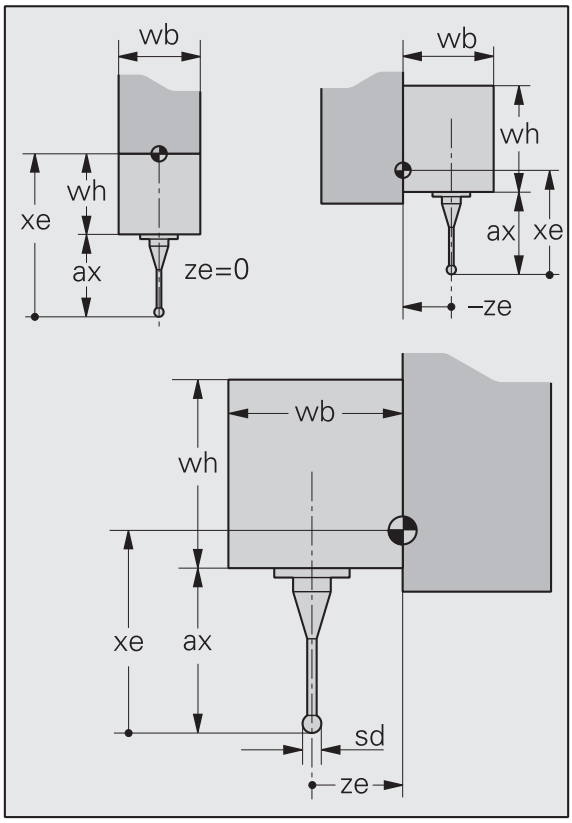
Additional information:

- Dialog box 3: see “Multipoint tools, tool life monitoring” on page 620
- see “Explanation of tool data” on page 622
- see “Tool holder, tool mount” on page 624

Parameters for workpiece handling systems and encoders

Example: Tool type 811

Parameters in dialog box 1	G	S	TP
ID: Tool ID number	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Available: Physical availability	•	–	–
Shank d (sd): Shank diameter	–	•	–
Multi. tool: Multipoint tool (see "Tool programming" on page 121) ■ **No:** No multipoint tool ■ **Main:** Primary cutting edge ■ **Secondary:** Secondary cutting edge	•	–	–
M ID: Identification number of the following cutter of a multipoint tool	•	–	–
T hold. DIN: Type of tool holder	–	•	–
T hold. heig. (wh): Height of tool holder	–	•	–
T hold. brea. (wb): Width of tool holder	–	•	–
Sali.lg.(ax): Salient length	–	•	–
Picture no.	–	•	–
Mount type			
Mag(azine) **code**: Not used at present			
Mag(azine) **attr**(ibute): Not used at present			



Multipoint tools, tool life monitoring

Turning tools with more than one (maximal 5) cutting edges are referred to as multipoint tools. In the tool database, each cutting edge is defined in a separate data record. In addition, a "closed chain" of all cutting edges of the multipoint tool is formed.

Define one cutting edge as primary cutting edge and the other ones as secondary cutting edges. In the tool list, only the primary cutting edge is defined.

Parameters in dialog box 3
Mag(azine) **code**: Not used at present
Mag(azine) **attr**(ibute): As of software version 625 952-05. If it is prepared by the machine tool builder, the parameter for special treatment of the tool during tool change can be used (example: for tool cleaning).
Multi. tool: Multipoint tool (see "Tool programming" on page 121) ■ **No:** No multipoint tool ■ **Main:** Primary cutting edge ■ **Secondary:** Secondary cutting edge

Parameters in dialog box 3
M ID: Identification number of the following cutter of a multipoint tool
Mon(itoring) **type** of tool life monitoring (see "Tool programming" on page 121) ■ None ■ Tool life monitoring ■ Quantity, monitoring for number of parts produced
Tool life total: Tool life of the cutting edge
Tool life rem.: Display of remaining tool life
Quantity total: Total quantity that can be produced by a tool
Quantity remaining: Display of remaining quantity
Reason for stoppage: ■ Tool life expired ■ Quantity reached ■ Tool life expired: ■ Determined through in-process measurement ■ Determined through post-process measurement ■ Tool wear determined through load monitoring: ■ Limit value 1 or 2 of "capacity" exceeded ■ Limit value of "work" exceeded



The tool life parameters are reset with a new cutting edge (see "Tool life management" on page 88).

Data input for multipoint tools

For the primary cutting edge:

- Parameter input (dialog boxes 1 and 2).
- Switch to dialog box 3 by pressing the "Page Up" key.
- "Multi. tool" input box: Enter the **Main (cutting edge)**.
- "M ID" input box: Enter the ID number of the **next secondary cutting edge**.
- Conclude the dialog box by pressing OK.

For each secondary cutting edge:

- Enter the ID number (ID number specified in "M-ID" of the previous cutting edge).
- Enter the remaining parameters (dialog boxes 1 and 2).
- Switch to dialog box 3 by pressing the "Page Up" key.
- "Multi. tool" input box: Enter the **Secondary (cutting edge)**.
- "M ID" input box: Enter the ID number of the next secondary cutting edge. When defining the last secondary cutting edge, set "M ID" to the primary cutting edge.
- Conclude the dialog box by pressing OK.



Make sure you define a "closed chain" in multipoint tools (primary cutting edge – secondary cutting edges – primary cutting edge).

Explanation of tool data



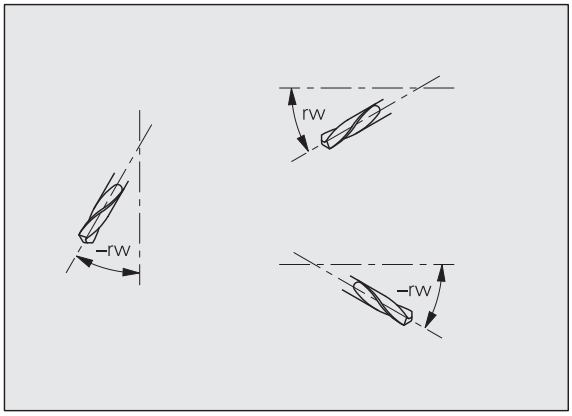
A ">>" after the input box means "fixed-word list." Select the tool parameter from the "fixed-word list" and use it as input.

Calling the fixed-word list: Position the cursor on the input box and press the ">>" soft key.

- **Tool ID number:** Each tool is assigned a unique ID (with up to 16 letters and numbers). The ID number must not start with an underline character.
- **Tool type:**
  - First, second digit: Type of tool
  - Third digit: tool position/main machining direction
- **Setting dimensions (xe, ye, ze):** Distance between tool reference point and the reference point of the tool carrier
  Available as of software version 625 952-05. Value range for setting dimensions:
  +/– 9 999.999 mm

- **Compensation values (DX, DY, DZ, DS):** Compensation for the wear of the cutting edge. For recessing and button tools, DS stands for the compensation value of the third side of the tool (away from the tool reference point).
- **Cutting length (sl):** Length of the cutting edge
  - The contour-based cycles check whether the tool can execute the required machining operation.
  - "sl" influences the tool selection of TURN PLUS.
  - "sl" is evaluated for the tool graphics and for depicting the cutting path of the tool.
- **Secondary machining direction (NBR):** Defines the directions in which the tool cuts in addition to the main machining direction.
  - The contour-based cycles check whether the tool can execute the required machining operation.
  - Influences the tool selection of TURN PLUS.
  - For NBR, the AWG uses the auxiliary feed rate (see "Technology Database" on page 641) and a reduced cutting depth (see machining parameter 4 – "SRF")
- **Direction of rotation:**
  - Defines the direction of spindle rotation for the tool.
  - Defines whether a driven tool is used.
  - The contour-based cycles check whether the tool can execute the required machining operation.
  - Influences the tool selection of TURN PLUS.
  - Defines the direction of spindle rotation in the AWG.
- **Width (dn):** Tool width from tool tip to shank back. "dn" is used for the tool graphics.
- (Physically) **Available**: Identifies an unavailable tool without deleting the database entry.
- **Design** as "left-hand or right-hand tool" – defines the position of the tool reference point. For neutral tools, the tool reference point is located on the left side.
- **Picture number:** Display the tool or only the cutting edge?
  - 0: Display the tool
  - –1: Display only the cutting edge of the tool
- TURN PLUS multiplies the following compensation values by the cutting values calculated from the technology database:
  - **CSP compensation:** Cutting speed
  - **FDR compensation:** Feed rate
  - **Depth compensation:** Cutting depth

- **Mount type:** If different tool mounts exist, the tool and the tool location must have the same mount type (see MP 511, ...).
  - Influences the tool selection and tool placement in TURN PLUS.
  - The "Set up tool table" functions check whether the tool can be used at the specified turret position.
- **Angle of orientation (rw):** Defines the deviation from the main machining direction in the mathematically positive direction of rotation (–90° < rw < +90°) – see illustration. TURN PLUS only uses drilling and milling tools machining in direction of or at right angles to the principal axis.
- **No of teeth:** Used for "G93 feed rate per tooth"
- **Salient length (ax):** For drilling and milling tools:
  - Axial tools: ax = distance between the tool reference point and the top edge of the tool holder
  - Radial tools: ax = distance between the tool reference point and the lower edge of the tool holder (also applies if the drill/mill is chucked)



Tool holder, tool mount

Tool holder

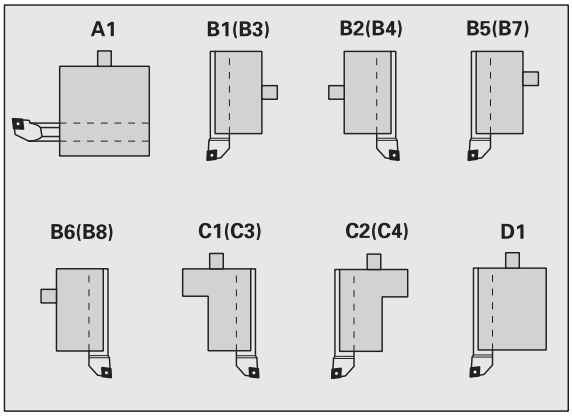
The graphic tool representation functions in the simulation and control graphics take account of the shape of the tool holder and the mounting position on the tool carrier. When the tool holder type is not specified, the CNC PILOT uses a simplified graphic representation.

Depending on the turret location, the CNC PILOT determines whether the holder is mounted in an axial or radial position or whether an adapter is used.

The CNC PILOT recognizes the following holders (designation of the standard holder as per DIN 69 880):

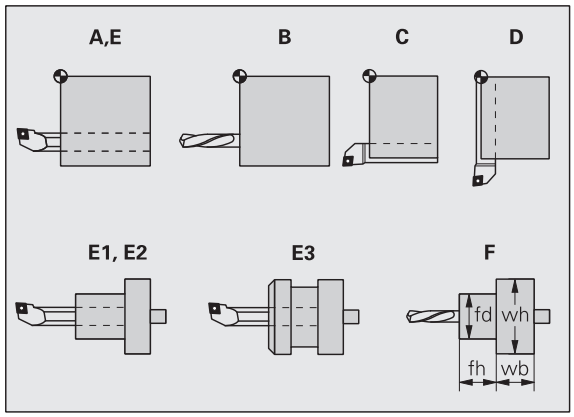
Holder group 1

- A1 boring bar holder
- B1 right-hand, short design
- B2 left-hand, short design
- B3 right-hand, short design, overhead
- B4 left-hand, short design, overhead
- B5 right-hand, long design
- B6 left-hand, long design
- B7 right-hand, long design, overhead
- B8 left-hand, long design, overhead
- C1 right-hand
- C2 left-hand
- C3 right-hand, overhead
- C4 left-hand, overhead
- D1 Multicarrier



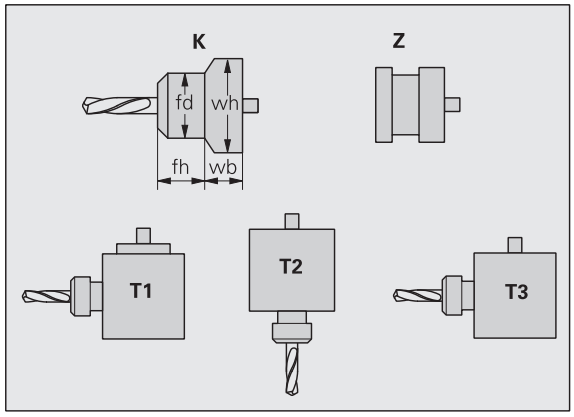
Holder group 2

- A boring bar holder
- B drill holder with coolant supply
- C square, longitudinal
- D square, transverse
- E face machining
- E1 U drill
- E2 cylindrical shank holder
- E3 Collet chuck holder
- F drill holder MK (Morse taper)



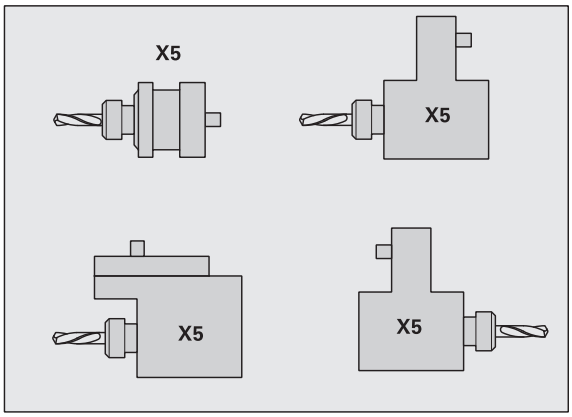
Holder group 3

- K Drill holder
- Z stop
- T1 driven, axial
- T2 driven, radial
- T3 boring bar holder



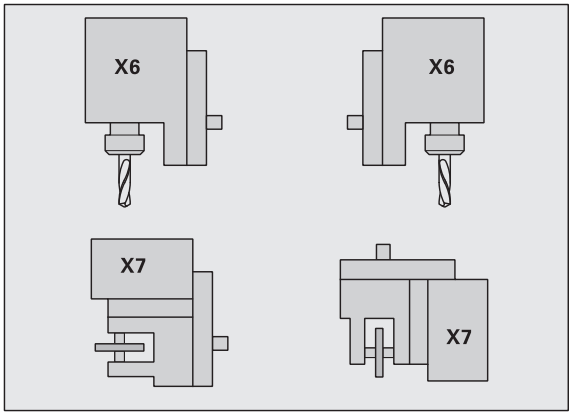
Holder group 4

- X5 driven, axial



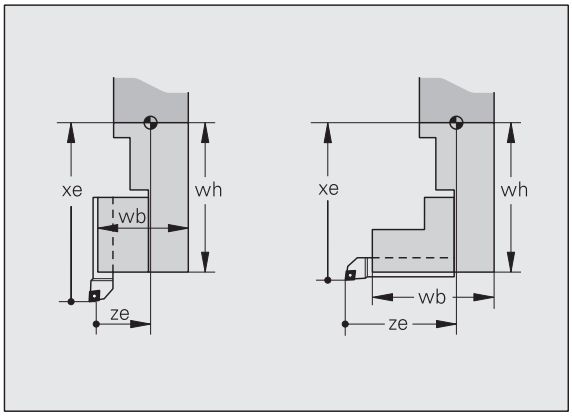
Holder group 5

- X6 driven, radial
- X7 driven, special holder



Adapter

When an adapter is used, the values entered for tool height (wh) and tool width (wb) refer to the height/width of the adapter and the holder.

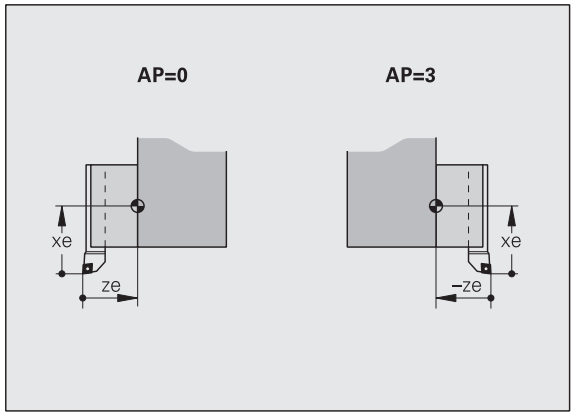


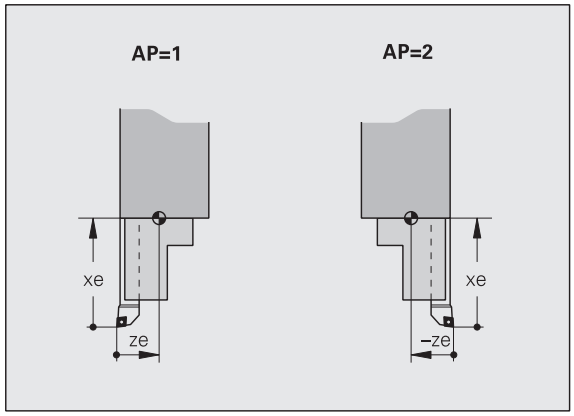
Tool mounting position

The mounting position is fixed by the machine tool builder (see MP 511, ...) Depending on the turret location, the CNC PILOT determines the tool mounting position:

- AP=0: axial mount – left turret side
- AP=1: radial mount – left turret side
- AP=2: radial mount – right turret side
- AP=3: axial mount – right turret side

When the tool is mounted radially in the middle of the turret, “AP=1” is used.





8.2 Database for Chucking Equipment

The CNC PILOT stores up to 999 chucking equipment descriptions which are managed via the chucking equipment editor. Chucking equipment is used in TURN PLUS mode and displayed in the simulation/control graphics. The chuck parameters can be omitted provided that you do not use TURN PLUS, or do not wish chucks to be displayed in the simulation graphics.

ID number: Each chuck has its own ID number (up to 16 numbers/ letters). The ID number must not start with with an underline character.

Chuck type: The chuck type defines the type of chuck/jaws.

Chucking equipment editor

The chucking equipment data contains information for representation in the simulation/control graphics, and further data for chuck selection in TURN PLUS.

Call the chucking equipment editor:

- „Select "**Chuck**(ing equipment)" in the Parameter mode.

Describing new chucking equipment ("New direct")

Select "New direct."

Enter the chuck type directly.

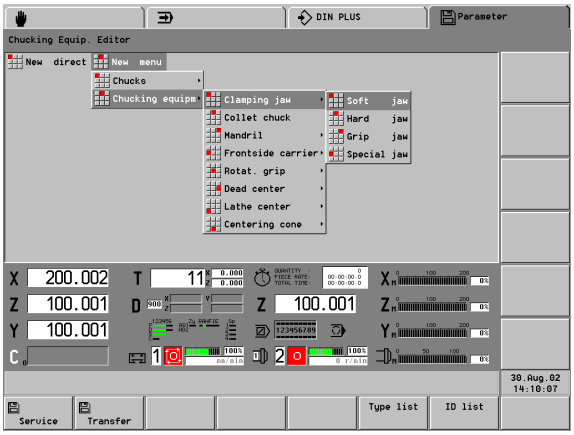
Enter the chucking equipment data.

Describing new chucking equipment ("New menu")

Select "New menu."

Select the chuck type from the submenus.

Enter the chucking equipment data.



Soft keys	
Service	Change to Service mode
Transfer	Change to Transfer mode

Chucking equipment lists

The CNC PILOT lists the chucking equipment according to identification numbers or chuck types. The chucking equipment list serves as starting point for editing, copying or deleting entries.

The header of the list indicates the entered mask, the number of chucks found, the number of chucks stored, and the maximum number of chucks.

Calling the chucking equipment list

Type list	The editor lists the entries sorted by chuck type.
ID list	The editor lists the entries sorted by ID number. Only those entries are displayed that correspond to the mask for ID numbers.

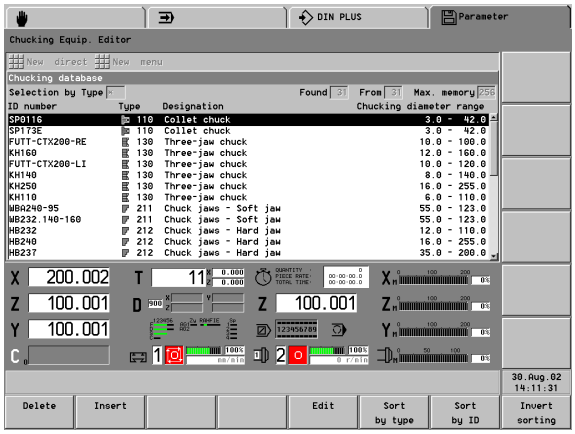
Mask for ID numbers:

- Enter part of the ID: Any desired (number of) characters can be specified at this position in the mask.
- "?": Any desired character can be specified at this position in the mask.

Editing the chucking equipment list

Place the cursor on the chucking equipment you want to edit.

Insert	Copy the entry (only chucking equipment of the same type).
Delete	Deleting an entry
Edit	Press the soft key or Enter key. The CNC PILOT presents the chucking equipment data for editing.



Soft keys

Delete	Delete a chuck entry
Insert	Copy chucking equipment entry
Edit	Editing a chuck entry
Sort by type	Sort the chucking equipment list by type
Sort by ID	Sort the chucking equipment list by ID number
Invert sorting	Reverse the sorting sequence

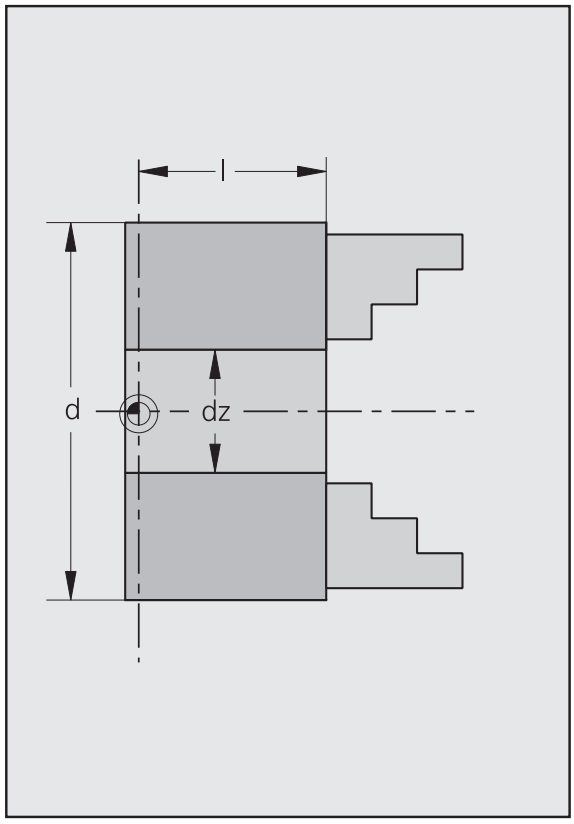
Chucking equipment data

Overview of chuck types			
Primary chucking equipment		**Chucking equipment**	Model
Chuck		Chuck jaws	21x
Chucking equipment		Collet chuck	220
Chuck	Model	Mandrel	23x
Collet chuck	110	Face driver	24x
Two-jaw chuck	120	Rotating gripper	25x
Three-jaw chuck	130	Dead center	26x
Four-jaw chuck	140	Lathe center	27x
Face chuck	150	Centering cone	28x
Special chuck	160		
Adapters for chuck type 21x		**Adapters for chuck types 23x..28x**	
Soft jaws	211	Cylindric chuck adapter	xx1
Hard jaws	212	Plain flange adapter	xx2
Grip jaw	213	Morse cone MK3	xx3
Special jaw	214	Morse cone MK4	xx4
		Morse cone MK5	xx5
		Morse cone MK6	xx6
		Other adapters	xx7

Chuck

Example: Three-jaw chuck (type 130)

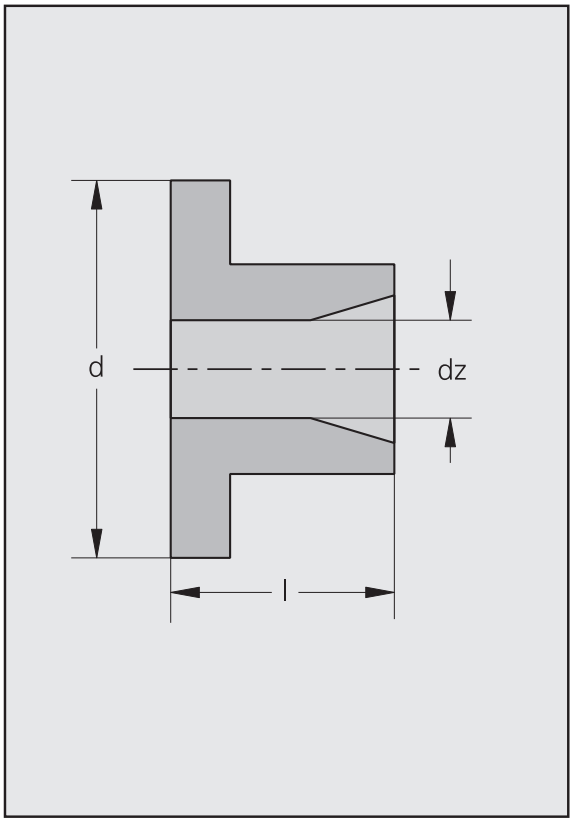
Parameters, chuck (type 1x0)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Jaw con.: Code for jaw adapter
d: Chuck diameter
l: Chuck length
max.cl.dia. (d1): Maximum clamping diameter
min.cl.dia. (d2): Minimum clamping diameter
dz: Centering diameter
max.speed: Maximum spindle speed (rpm)



Code for jaw adapter: When specific combinations of chuck and jaws are required, enter the appropriate jaw adapter code. For both the chuck and the required chuck jaws, enter the same code.

Jaw con.=0: All chuck jaws are permitted.

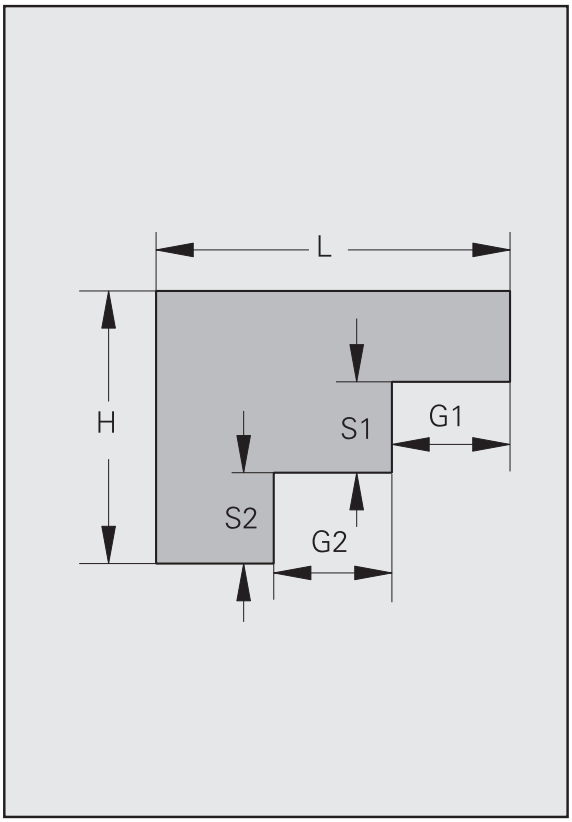
Example: Collet chuck (type 110)



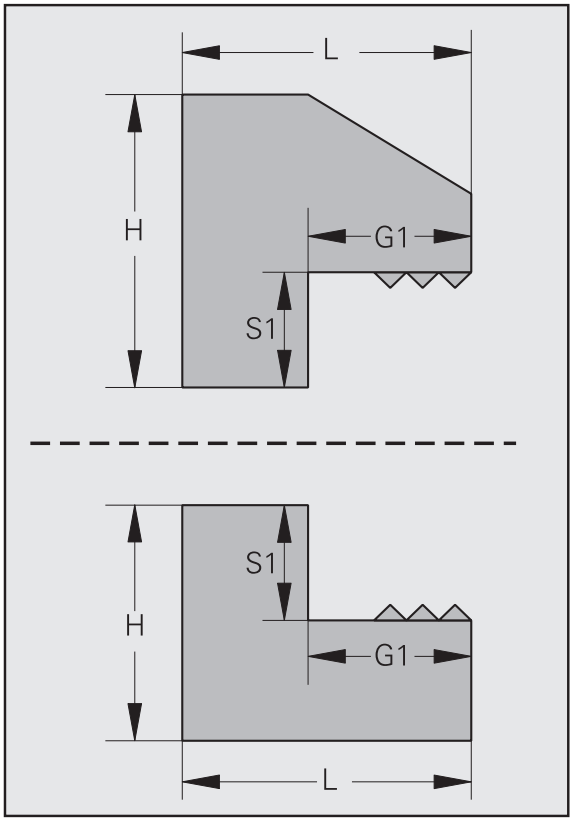
Chuck jaws

Example: Chuck jaw (type 211)

Parameters, chuck jaws (type 21x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Jaw con.: Code for jaw adapter – must correspond to the jaw adapter code of the chuck
L: Width of jaws
H: Height of jaws
G1: Dimension, step 1 in Z direction
G2: Dimension, step 2 in Z direction
S1: Dimension, step 1 in X direction
S2: Dimension, step 2 in X direction
min.cl.dia. (d2): Minimum clamping diameter
max.cl.dia. (d1): Maximum clamping diameter



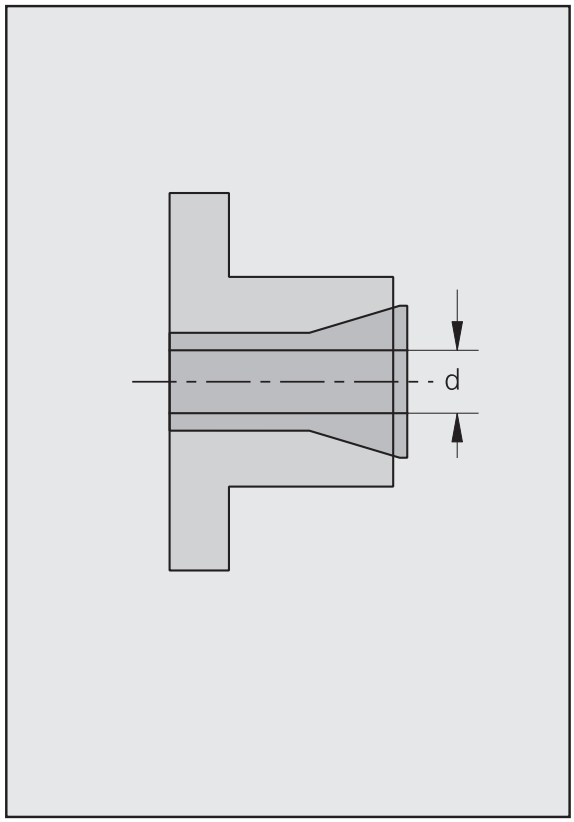
Example: Grip jaw (type 213)



Collet chuck

Example: Collet chuck (type 220)

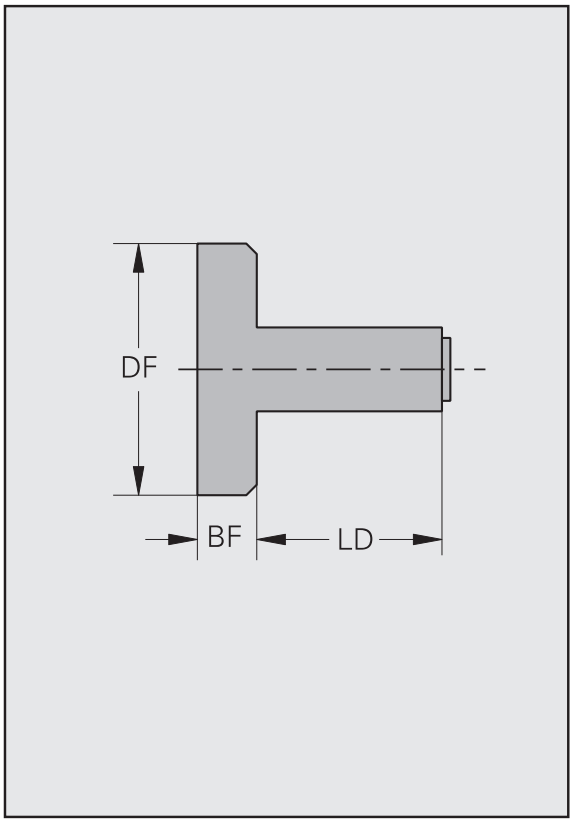
Parameters, collet chuck (type 220)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
d: Diameter of collet chuck



Mandrel

Example: Mandrel (type 231)

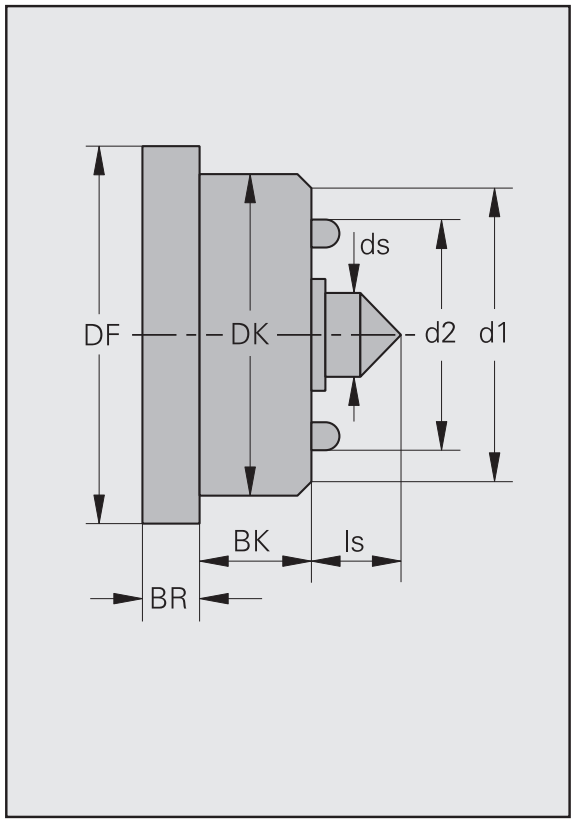
Parameters, mandrel (type 23x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Mandril length
LD: Total length
DF: Flange diameter
BF: Flange width
max.cl.dia.: Maximum clamping diameter
min.cl.dia.: Minimum clamping diameter



Face driver

Example: Face driver (type 241)

Parameters, face driver (type 24x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
ds: Tip diameter
ls: Tip length
DK: Body diameter
BK: Body width
DF: Flange diameter
BR: Flange width
d1: Maximum clamping diameter
d2: Minimum clamping diameter



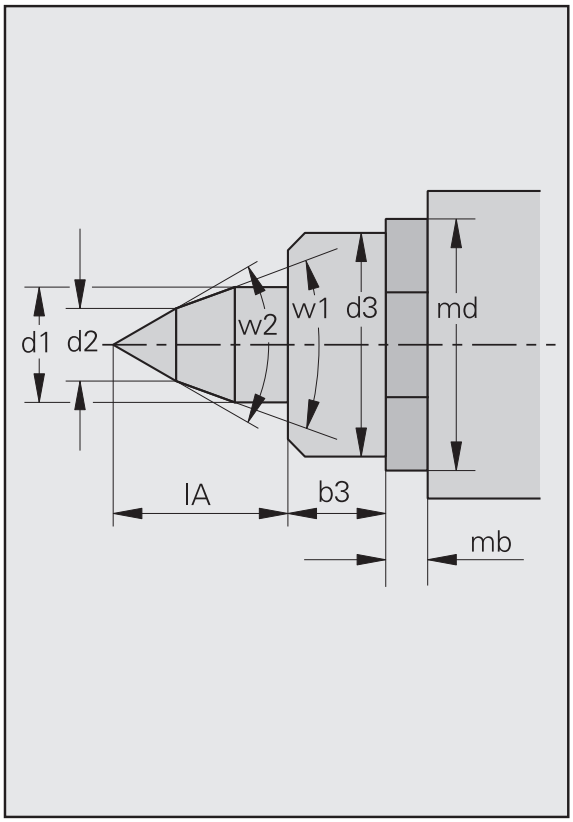
Rotating gripper

Parameters, rotating gripper (type 25x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Nom.dia.: Diameter of rotating gripper
Length: Length of rotating gripper
d1: Maximum clamping diameter
d2: Minimum clamping diameter

Dead center

Example: Dead center (type 261)

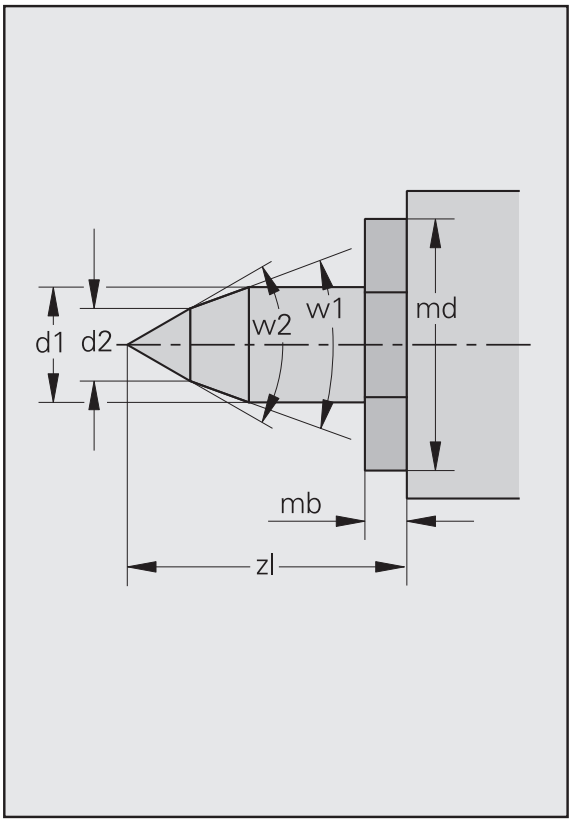
Parameters, dead center (type 26x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
w1: Point angle 1
w2: Point angle 2
d1: Diameter 1
d2: Diameter 2
lA: Length of conical part
d3: Diameter of the dead center sleeve
b3: Width of the dead center sleeve
md: Circumscription diameter of the forcing nut
mb: Width of the forcing nut



Lathe center

Example: Lathe center (type 271)

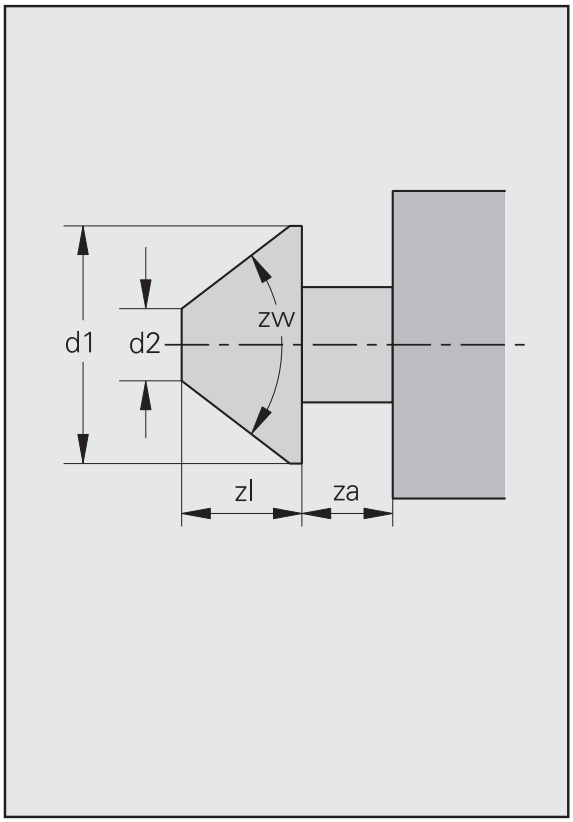
Parameters, lathe center (type 27x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
w1: Point angle 1
w2: Point angle 2
d1: Diameter 1
d2: Diameter 2
zl: Length of lathe center
md: Circumscription diameter of the forcing nut
mb: Width of the forcing nut



Centering cone

Example: Centering cone (type 281)

Parameters, centering cone (type 26x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
zw: Centering cone angle
za: Length from centering taper tip to the sleeve
d1: Diameter 1
d2: Diameter 2
zl: Length of centering cone



8.3 Technology Database

The CNC PILOT saves the technology data (cutting values) in a three-dimensional table depending on the following information:

- Material (of the workpiece)
- Cutting material (material of the tool's cutting edge)
- Machining operation

The types of machining are specified. Define the workpiece material and cutting material by fixed-word list and assign them to the table (see figure).

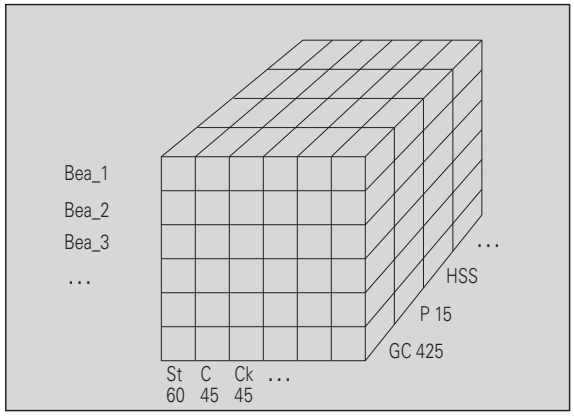
The cutting data is managed in the technology editor.

The TURN PLUS working plan generation uses the technology data. In addition, you can use the database for saving your "own" cutting data.



The data defined in the fixed-word lists for the workpiece material and cutting material must be aligned with the cutting data entered.

If you change the fixed-word list of workpiece/cutting material, the cutting data will **not** be adjusted automatically. In this case, you also need to change the cutting data to ensure correct technology data.



Explanations

Machining method:

- bea_1: Roughing
- bea_2: Finishing
- bea_3: Recessing
- etc.

Cutting material (defined by fixed-word list):

- Gc425
- P15
- HSS
- etc.

Material (defined by fixed-word list):

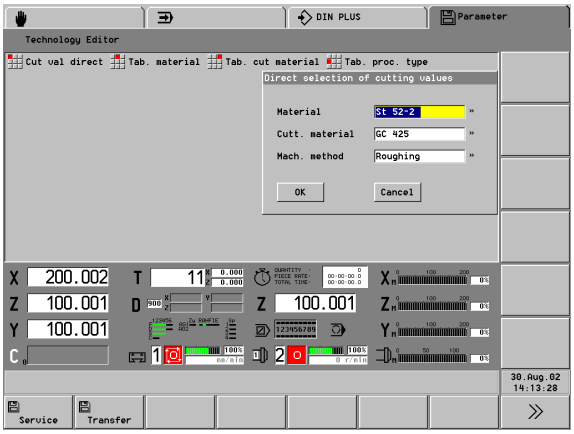
- St60
- C45
- Ck45
- etc.

Editing the technology data

The technology database contains the following data:

- **Specific cutting effort** for the material: The parameter is for information only; it is not evaluated.
- **Cutting speed**
- **Main feed rate [mm/rev]** for the main machining direction
- **Auxiliary feed rate [mm/rev]** for the secondary machining direction
- **Depth of cut**
- With/without **coolant**: The automatic working plan generation (AWG) uses this parameter to determine whether coolant is used.

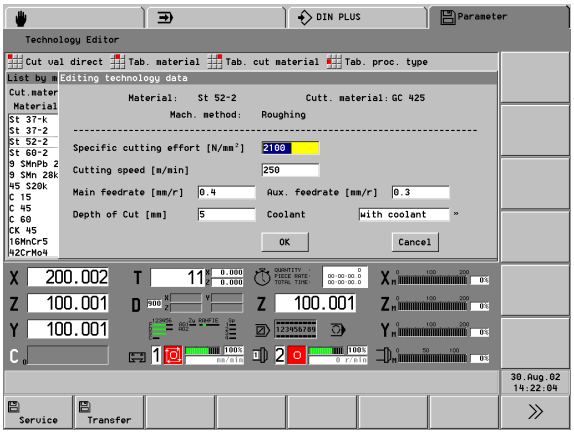
TURN PLUS multiplies the cutting values by the compensation factors (CSP, FDR, DEPTH comp) defined for the tools (see “Explanation of tool data” on page 622).



Editing the technology data

Select “Cut val direct.” The CNC PILOT opens the “Direct selection of cutting values” dialog box.

Specify the material, cutting material and machining method. The CNC PILOT opens the “Editing technology data” dialog box and presents the data for editing.



Cutting-value tables

Call the technology editor:

▶ Select “Tech(nology data)” in the Parameter mode.

Calling the cutting-value tables

Select “Tab. material.” The “List by materials” dialog box appears.

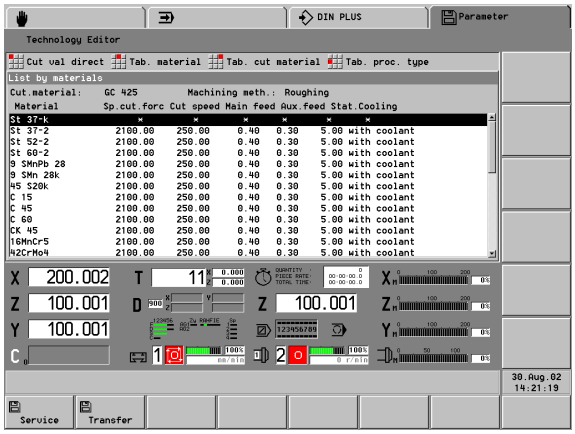
Specify the machining method and cutting material. The CNC PILOT lists the technology data, sorted by material.

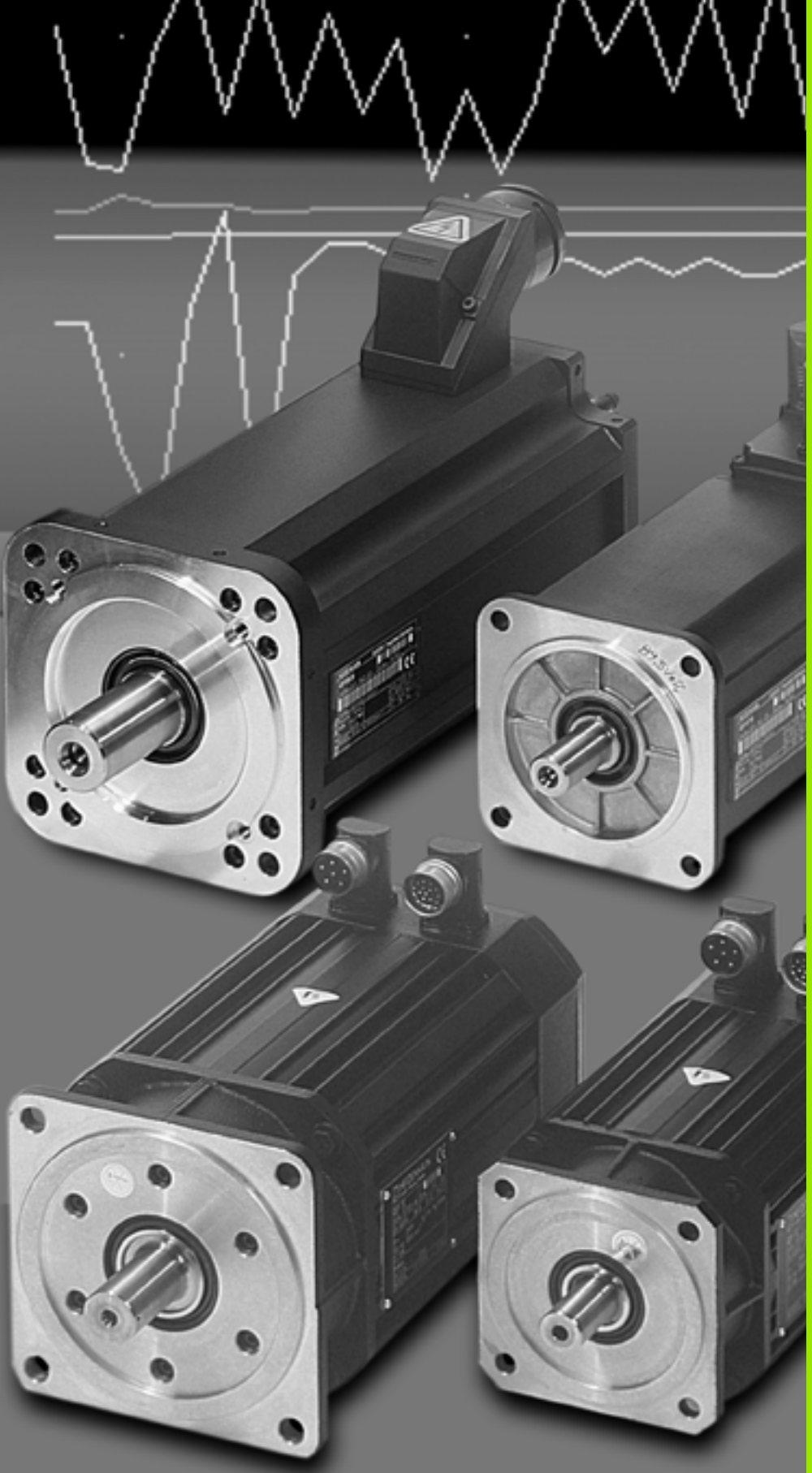
Select “Tab. cut material.” The “List by cutting materials” dialog box appears.

Specify the material and machining method. The CNC PILOT lists the technology data, sorted by cutting material.

Select “Tab. proc. type” (machining method). The “List by machining method” dialog box appears.

Specify the material and cutting material. The CNC PILOT lists the technology data, sorted by machining method.


DIN PLUS
Parameter
Technology Editor
Cut val direct
Tab. material
Tab. cut material
Tab. proc. type
List by materials
Cut.material: GC 425 Machining meth.: Roughing
Material Sp.cut.forc Cut speed Main feed Aux.feed Stat.Cooling
St 37-k
St 37-2 2100.00 250.00 0.40 0.30 5.00 with coolant
St 52-2 2100.00 250.00 0.40 0.30 5.00 with coolant
St 60-2 2100.00 250.00 0.40 0.30 5.00 with coolant
9 SMnPb 28 2100.00 250.00 0.40 0.30 5.00 with coolant
9 SMn 28k 2100.00 250.00 0.40 0.30 5.00 with coolant
45 S20k 2100.00 250.00 0.40 0.30 5.00 with coolant
C 15 2100.00 250.00 0.40 0.30 5.00 with coolant
C 45 2100.00 250.00 0.40 0.30 5.00 with coolant
C 60 2100.00 250.00 0.40 0.30 5.00 with coolant
CK 45 2100.00 250.00 0.40 0.30 5.00 with coolant
16MnCr5 2100.00 250.00 0.40 0.30 5.00 with coolant
42CrMo4 2100.00 250.00 0.40 0.30 5.00 with coolant
X 200.002
Z 100.001
Y 100.001
C
T 11
D
Z 100.001
30.Aug.02
14:21:19
Service
Transfer



9

Service and Diagnosis

9.1 The Service Mode of Operation

The Service mode of operation features:

- **Service functions:** User registration and user management, language switching and different system settings
- **Diagnostic functions:** Functions for system inspection and support when searching for errors
- **Maintenance system:** Reminds the machine user about necessary maintenance and repair tasks.



Some service and diagnostic functions, such as the oscilloscope or the logic analyzer, are not accessible (reserved for service and commissioning personnel).

9.2 Service Functions

Access authorization

Functions such as editing important parameters are reserved for privileged users. The control permits access when the correct **password** is entered. The access authorization remains in effect until the user logs off or until a second user logs on correctly.

The password consists of a 4-digit number. It is entered "masked" (not visible).

The CNC PILOT differentiates between:

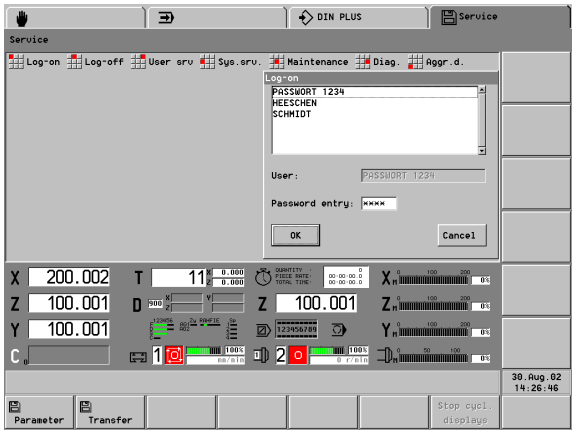
- Without protection class
- NC programmers
- System managers
- Service personnel (of the machine tool builder)

"Log-on" menu item: For user logon, select your name from the list of users and enter your password.

"Log-off" menu item: The CNC PILOT does not support automatic log-off. Thus it is important that the user log off to prevent illegal access by someone else.

"User srv" menu item: User service functions are only available after log-on as a system manager.

- **Enter user:** Enter the name of the new user, assign a password and user class. Precondition: You are logged on as a system manager.
- **Cancel user:** Select the user to be deleted and confirm with OK.
- **Change password:** Every user can change his or her assigned password. To safeguard against misuse, the user must first enter the "old" password before assigning a new one.



- The CNC PILOT is shipped with the user name "Password 1234" and the password "1234" (user class: system manager). To enter further users, log on with the user name "Password 1234." You should then delete the user "Password 1234".
- The CNC PILOT prevents the deletion of the "last" system manager. Make sure you remember the password, however.

System service

"Sys.srv." (System service) drop-down menu

- **Date/Time:** Error messages are recorded together with the date and time they occurred. Since all errors are stored in a log file for a long period of time, you should always ensure that the date and time are correctly set. These data facilitate the fault diagnosis in case you require field service.
- **Language switching:** Select the language with the soft key ">>" and press "OK". The selected screen language becomes effective as soon as you restart the CNC PILOT.
- **FWL-editing – Language dependent:** Currently not used.
- **FWL-editing – Language independent:** Editing the fixed-word lists for workpiece material, cutting material and fits (see "Fixed-word lists" on page 649).
- **Aux. images ON/OFF:** When "Aux. images ON" is active, the graphic support windows of Machine mode are not displayed.
- **Editing switch ON/OFF:** This function protects the following operating modes from unauthorized access. When "Editing switch ON" is active, these menu items can only be selected if you are logged on as an NC programmer (or higher):
  - DIN PLUS
  - TURN PLUS
  - Parameter

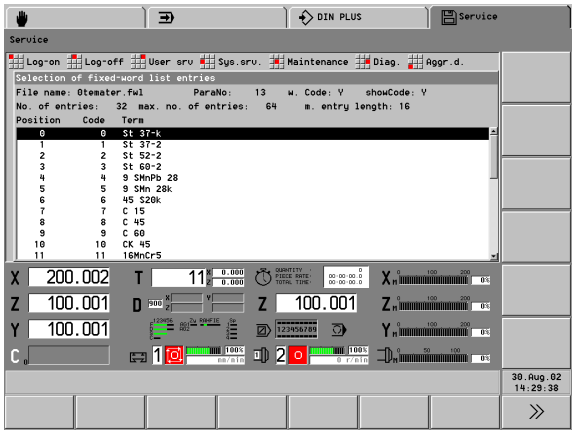
Fixed-word lists

Material/Cutting material: The CNC PILOT manages the designations of workpiece materials and cutting materials in fixed-word lists. Use them to adapt the technology database to your requirements (see "Technology Database" on page 641).

Fit: The parameter "Fit" is included for the delta drill and reamer tools. Specify the desired fit qualities in the "0WZPASSU" fixed-word list.

Note on editing fixed-word lists:

- Maximal 64 entries
- **Code**
  - Numbers from 0 to 63
  - Do not assign the same code twice.
- **Designation**
  - Maximal 16 characters



Editing a fixed-word list

Select "Sys.Srv. > FWL-editing > Language independent". The CNC PILOT opens the "Selection of fixed-word list entries" dialog box.

Select one of the following files:

"0TEMATER" (workpiece material)

"0TESTOFF" (cutting material)

"0WZPASSU" (fit quality)

Edit entry:

Select the entry you want to edit. Press ENTER.

Change the code and/or designation.

Press OK The CNC PILOT saves the data.

New entry:

Ins — Open the "Editing fixed-word lists" dialog box.

Enter a code and designation.

Press OK. The CNC PILOT saves the data.

9.3 Maintenance System

The CNC PILOT reminds the machine user of the required maintenance and repair tasks. Each action is described in brief form (assembly, maintenance interval, responsible person, etc.). This information is shown in the "maintenance and repair jobs" list. A comprehensive description of the maintenance action is provided if desired.

As soon as you acknowledge a completed maintenance measure, the maintenance interval restarts. The CNC PILOT saves the time of acknowledgment together with the nominal date in a log file. Service personnel can evaluate the acknowledgment log files. You can see at least the last 10 acknowledgments.

Maintenance status: The "traffic light" next to the date/time field displays the maintenance status. The status with the highest priority is displayed (red before yellow, yellow before green).

- Green: No maintenance measures necessary.
- Yellow: At least one maintenance measure will soon be due.
- Red: At least one maintenance measure is due or overdue.

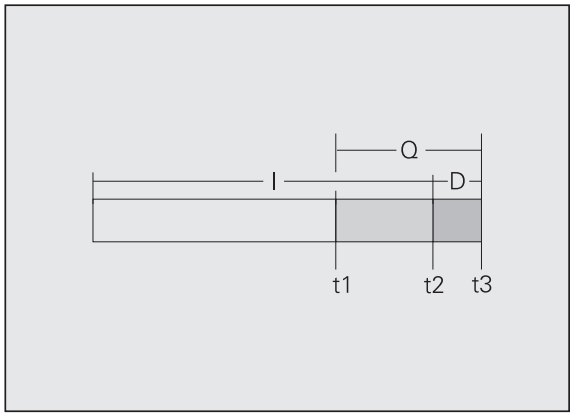


- **Prerequisite:** The machine tool builder must enter the required actions and provide complete descriptions of them.
- All status changes including the acknowledgment of maintenance measures are reported to the PLC. Refer to the machine manual to see whether further consequences must be made from due or overdue maintenance actions.

Maintenance dates and intervals

Dates and intervals (see illustration):

- **I – interval**: Time period of the maintenance interval, specified by the machine manufacturer. During the on-time of the control, the current maintenance interval is continuously reduced. The maintenance system displays the remaining time in the "**When**" column.
- **D – duration**: Time period between "due" and "overdue," specified by the machine manufacturer.
- **Q—acknowledgment time period:** In this period the maintenance measures must be performed and acknowledged.
- **t1 – time point "Maintenance measure will soon be due":**
  - Starting from this time point, the maintenance measures **can** be performed and acknowledged.
  - The status is marked in yellow.
  - Calculation: t1 = early warning * interval / 100
- **t2 – time point "Maintenance measure is due":**
  - Starting from this time point, the maintenance measures **should** be performed and acknowledged.
  - The status is marked in red.
  - Calculation: t2 = interval
- **t3 – time point "Maintenance measure is overdue":**
  - At this time point, the maintenance measures are **overdue**.
  - The status stays on red.
  - Calculation: t3 = interval + duration



Explanations:

I:	Interval
D:	Duration
Q:	Acknowledgment time period
t1:	Maintenance action will soon be due
t2:	Maintenance action is due
t3:	Maintenance action is overdue

Displaying maintenance actions

Information on maintenance actions

Call the maintenance system:

- Select “Maintenance” in the Service mode. The maintenance system displays the “Maintenance and repair actions” list.



- Change to part 2 of the list.

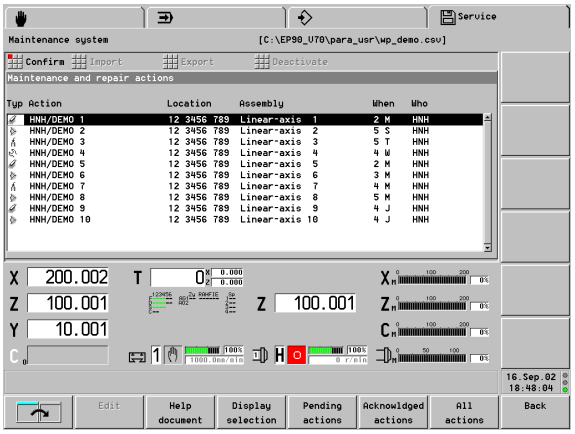


- Change to part 1 of the list.

- The vertical arrow keys and the Page Up/Dn keys move the cursor within the list.

ESC

- Return to Service mode.



Call the lists of maintenance actions:

Pending actions

- Call the list of pending, current and overdue maintenance actions, or

All actions

- Call the list of all maintenance actions.

Display additional information:

- Place the cursor on the required maintenance action.
- Press ENTER. The maintenance system opens the “Read action” dialog box containing the parameters of the action, or

Help document

- Call a detailed description of the maintenance action.

ESC

- Return to the list of maintenance actions.

Type of maintenance action	Unit of time	
Cleaning	M / M:	Minutes
Inspection	S / H:	Hours
Maintenance	T / D:	Days
Repair	W / W:	Weeks
	J / Y:	Years

The entries in the **Maintenance actions** list have the following meanings:

- **Type**: See the "Type of maintenance action" table.
  The **status** is indicated by the background color:
  - No color: no maintenance action required
  - Yellow: maintenance action will soon be due
  - Red: maintenance action is due or overdue
- **Location**: Location of the assembly
- **Assembly**: Designation of the assembly
- **When**: Time remaining until maintenance action is due (= remaining time of the maintenance interval)
- **Duration**: Time period between "maintenance is due" and "maintenance is overdue"
- **Who**: Person responsible for performing the action
- **Interval**: Time remaining in the maintenance interval
- **Early warning**: Defines the time point of the "action is soon due" status (relative to the maintenance interval)
- **Documentation, reference doc:**
  - For an existing entry: Press the "Help document" soft key to display a detailed description of the maintenance action.
  - No entry: No description is available for the maintenance action.



- "–" before the symbol: The maintenance system is deactivated.
- Parts of a time unit are given as decimal numbers. Example: 1.5 S = 1 hour and 30 minutes.

Special maintenance action lists

Call lists according to the “Type” or “Status” of maintenance actions:

Display selection

▶ Switch to the soft-key row “Type/status of actions”.



▶ Call the “All repair actions” list or another special list (see soft-key table).

Back

▶ Return to the general maintenance system.

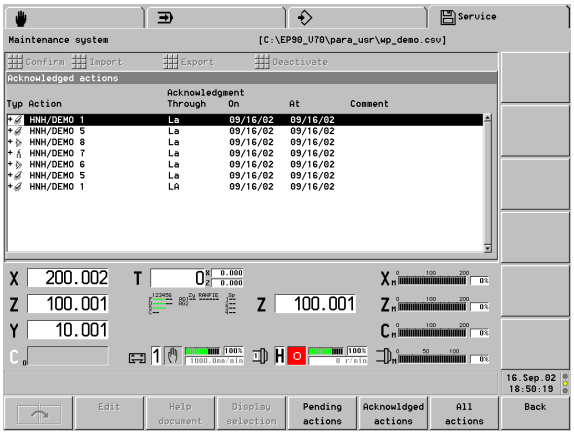
Call the list of acknowledged maintenance actions:

Acknowldged actions

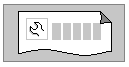
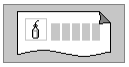
▶ Call the “Acknowledged actions” list.

The entries in the **Acknowledged actions** list have the following meanings:

- **Type**:
  - Symbol: see the “Type of maintenance action” table
  - “+”: action acknowledged
- **Action**: Designation of the maintenance action
- **Acknowledgment Through**: Name of the acknowledger
  Acknowledgment On: Date of the acknowledgment
- **At**: Time point “Maintenance action is due” (t2)
- **Comment** of the acknowledger



“Type of maintenance actions” soft keys

	All repair actions
	All maintenance actions
	All inspection actions
	All cleaning actions

“Status of maintenance actions” soft keys

	Pending maintenance actions
	Due and overdue maintenance actions

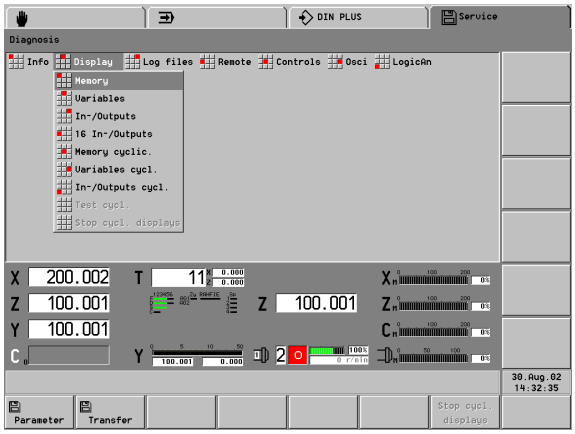
9.4 Diagnosis

Information and display

Call the Diagnosis submenu:

▶ Select “Diag(nosis)” in the Service - Maintenance mode.

ESC ▶ Return to Service mode.



The “Diagnosis” submenu provides information, test and control functions that help you with troubleshooting.

“Info” menu item: Provides information on the software modules being used.

As of software version 625 952-02:

Information about OEM data is also displayed, if available.

“Display” drop-down menu

- **Memory:** Reserved for service personnel
- **Variables:** Displays the “variable dump” (currently containing approx. 500 V variables)
  - “---”: the variable is not initialized
  - “???”: the variable is not available
- **In-/Outputs:** Shows the momentary status of all inputs and outputs
- **16 In-/Outputs:** Select up to 16 inputs/outputs in the “Select I/Os for display” dialog box. After the dialog box has been concluded, the CNC PILOT displays the status of these inputs/outputs. Each change of status is displayed immediately.
  To exit the display function, press the ESC key.
- **Memory cyclic.:** Reserved for service personnel
- **Variables cycl.:** Select a V variable. The CNC PILOT displays the value. Each change in value is displayed immediately.
- **In-/Outputs cycl.:** Select an I/O position. The CNC PILOT displays the status. Each change of status is displayed immediately.



The cyclic display window is superimposed on part of the machine display. To stop the cyclic display, select the “Display > Stop cycl. displays” menu item or press the “Stop cycl. displays” soft key.

Log files and network settings

"Log files" drop-down menu: Errors, system events, and data exchange between different system components are recorded in log files.

- **Display error log:** Displays the most recent error message. To view further entries, press the PgUp/PgDn keys.
- **Save error log file:** Makes a copy of the error log file (file name: error.log; Directory: Para_Usr). Existing "error.log" files are overwritten.
- **Save Ipo trace:** Saves information on the most recent interpolator functions (file names: IPOMakro.cxx, IPOBewbe.cxx, IPOAxCMD.cxx – xx: 00 to 99; directory: Data).

"Remote" drop-down menu: The Remote functions support **remote diagnosis**. For information on the Remote functions, refer to your machine manufacturer.

"Controls" drop-down menu

- **Hardware – System info**: Supplies you with information on the hardware components used.
- **Options**: Supplies you with an overview of the options that are available and installed on the CNC PILOT.
- **Network – Settings**: This menu item calls the "Network" WINDOWS dialog box. The CNC PILOT is entered as a "Client for Microsoft Networks." Details on installing and configuring networks are available in the documentation or in the on-line help function of Windows.
- **Network – Enabling password** (this function is only available on systems based on Windows 98): Assign individual passwords for read and write access. The passwords are effective for all shared directories (see "Enabling, file types" on page 668).
  The "Enabling names" listed in the "Enabling password" dialog box are for your information. Entries can only be made in the "Read password" and "Write password" input fields. The password is entered "masked" (not visible).

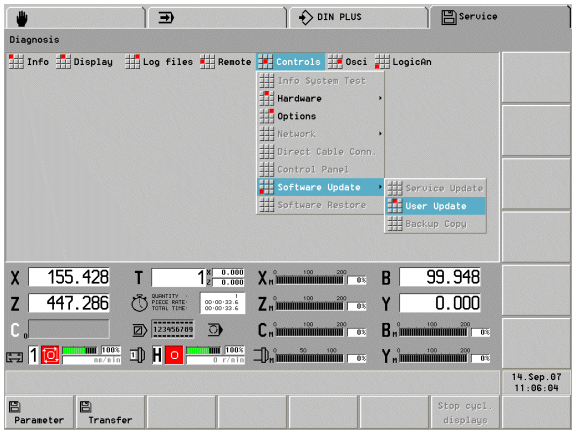
Menu items "Osci(loscope), LogicAn(alyzer)": Reserved for service personnel

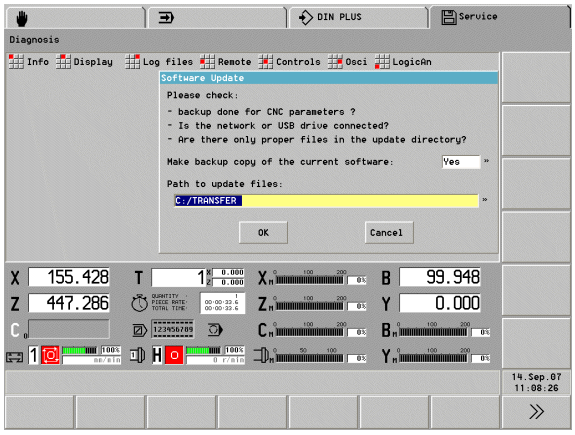
Software update

A software update provides new system functions or installs bug fixes released by HEIDENHAIN.

To update the software, proceed as follows:

- Log on in the System Manager user class.
- Select “Controls > Software Update > User Update“ in the Diagnosis menu. The CNC PILOT opens the “Software Update” dialog box.
- In this dialog box, the CNC PILOT offers to make a backup copy of the current software. HEIDENHAIN recommends making this backup. In the “Path to update files” input field, enter the path to the connected PC or storage medium.
- The CNC PILOT backs up the data and then imports the update files.
- Wait until the software update is complete. Then switch the CNC PILOT off and on again.
- Check the CNC PILOT.





In a backup, the CNC PILOT writes the complete software, including parameters, operating-resource data, NC programs, etc., to the “CNC_Save” directory. Any old backup files are deleted.



10

Transfer

10.1 The Transfer Mode of Operation

The Transfer mode is used for **data backup** and **data exchange** with other data processing systems. The transferred files contain NC programs (DIN PLUS or TURN PLUS), *DXF files, parameter files, or files with information for service personnel (oscilloscope data, log files, etc.).

The Transfer mode also includes organizational functions such as copying, deletion, renaming, etc.

Data exchange with DataPilot: HEIDENHAIN offers the PC program package DataPilot 4290 to complement the CNC PILOT machine control. DataPilot has the same programming and testing function as the control. This means that you can create TURN PLUS and DIN PLUS programs on the PC, test it with program simulation and transfer it to the machine control.

Data backup: HEIDENHAIN recommends saving your CNC PILOT programs on a PC in regular intervals.

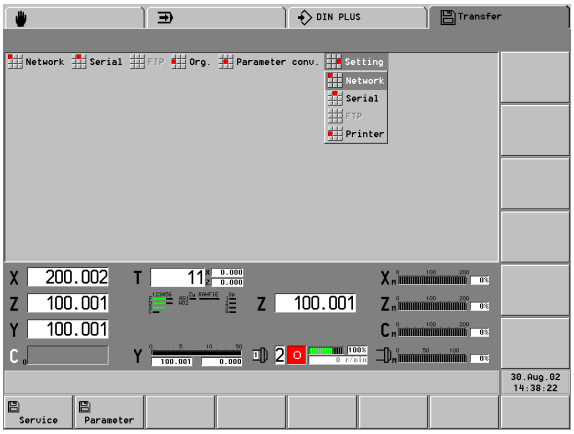
Since the parameters are not changed very often, however, you only need to back up the parameters from time to time, as required (see “Parameters and Operating Resources” on page 674).

Data backup systems: DataPilot can also be used for backing up NC programs created in the control. Of course, you can also use alternative operating system functions provided on your PC, or other programs available on the market for data backup.

Printer: With the organization functions you can prepare DIN PLUS programs and parameter/operating resource data for printout. The page format in the CNC PILOT defaults to size A4. You can print directly from DataPilot.

TURN PLUS programs cannot be printed out.

- TURN PLUS files are only processed by the CNC PILOT or DataPilot. They are not “readable”.
- Service files support troubleshooting. Theses files are normally transferred and evaluated by service personnel.



Soft keys	
Service	Change to Service mode
Parameter	Change to Parameter mode

Function overview of the Transfer mode of operation:

- **Network:** Activates the WINDOWS network and shows the "masked" files of the CNC PILOT and the communications partner.
- **Serial:** Activates serial data transmission and displays the "masked" files of the CNC PILOT.
- **FTP:** Activates the FTP network and shows the "masked" files of the CNC PILOT and the communications partner.
- **USB storage media:** The CNC PILOT supports Windows XP compatible USB mass storage devices.
- **Organization:** Management of local files.
- **Parameter conv(ersion):** Converts parameters/operating resources from "internal format" to ASCII format – or vice versa; prepares data backup; downloads saved data.
- **Setting:** Setting the network, FTP, serial interfaces or printer parameters.

Overview of data transfer methods

The CNC PILOT uses the Windows XPe operating system. Network communication is based on operating system functions. The network is therefore configured in Windows.

Interfaces: HEIDENHAIN recommends data transfer over an Ethernet interface. That guarantees high transmission speed, high reliability, and convenient operation. The USB interface is another reliable and easy-to-use way to transfer data, if suitable storage media are used. Data transmission via serial interface is also possible.

- **WINDOWS networks** (Ethernet interface): For integrating your lathe in a LAN network. The CNC PILOT supports the networks commonly used under WINDOWS.
  - The CNC PILOT allows you to send/receive files.
  - Other network users have read- and write-access to shared directories, regardless of the activities of the CNC PILOT.
  - As a rule, the CNC PILOT logs on to the network upon system startup and remains online until shutdown.
- **FTP – File Transfer Protocol** (Ethernet interface): For integrating your lathe in a LAN network. This requires an FTP server installed on the host computer.
  - The CNC PILOT allows you to send/receive files.
  - The CNC PILOT has no server functions. This means that other computer systems in the network cannot access CNC PILOT files.

- **USB interfaces:** The CNC PILOT is prepared for the connection of standard storage media with USB interface.
- **Serial:** You transfer program or parameter files via serial interface – without protocol. Ensure that your communications partner complies with the defined interface parameters (baud rate, word length, etc.).
- **Printer:** The CNC PILOT does not control the printer directly. Configure the CNC PILOT to output the print data as a file (see "General control parameters" on page 577). You can then print the data from this file.

USB storage media: The CNC PILOT identifies USB devices automatically. It also detects when a USB device is removed. The USB storage media are usually assigned the drive letter "D:". Devices other than USB storage media should be connected only with the permission of HEIDENHAIN.

Do not remove a USB device until data transfer for this device is completed.



- HEIDENHAIN recommends connecting or removing USB devices when the control is running. Because the first-time registration of a USB device requires a lot of computing power, you should only connect a new device when the machine is idle.
- It may happen that a USB device is not correctly read/written to, for example when the cable between the control panel and the main computer is very long. In this case, use a different USB device or connect the USB device directly to the control.

Configuring Windows networks



HEIDENHAIN recommends configuration of the Windows networks by the authorized personnel of your machine manufacturer.

Configuring a network

As of software version 625 952-04:

You configure the network and edit the settings in the Windows dialog box. To open the dialog box, select

- Diagnosis > Controls > Network > Settings

Activating/deactivating a network

As of software version 625 952-04:

The CNC PILOT activates/deactivates the network when you select the following menu items:

- Diagnosis > Controls > Network > ...
- „... > Network on: The network is activated
- „... > Network off: The network is deactivated

Logon as Windows users

For all other settings such as changing the computer name, etc. you need to log on as a Windows user, as described below.

The network is configured in Windows. At system start, Windows starts up with the Windows user group "CNCUser" enabled, but it remains in the background. The control software also starts up. The “Windows key” and the Windows key combinations ALT+TAB and CTRL+ESC have no function.

To enable the Windows key combinations, log on as **SERVICE-KEYBOARD** (Service operating mode / Logon). You can find the "SERVICE-KEYBOARD" user in the extended name list when you log on.

Log on in the Service-Keyboard class:

- Select “Log-on” in the Service mode.
- Choose any user.
- Enter 0.37 instead of the password. The CNC PILOT switches to the extended name list.
- Select SERVICE-KEYBOARD and enter the password 1306.

Call the Security window:

- Press CTRL+ALT+DEL. Windows opens the Security window.
- Use Log-off to log the current Windows user off.
- Log on with another Windows user name (with network configuration privileges, for example).

If you have to restart the operating system after changing operating-system settings, for example, you should shut down the system and switch off the control before restarting.



- As soon as you have activated Windows, the control screen is hidden. HEIDENHAIN therefore recommends that you do not activate Windows when automatic mode is active.
- Do **not** use the automatic restart in Windows.

User logon: You can only configure the network if you are logged on with the correct Windows user group. You log on in the required user group by entering the corresponding password. The meaning of the Windows passwords is different from the meaning of the passwords for control functions.

HEIDENHAIN has already installed the following users:

User name	User group	Password	Description
CNCUser	Users	–	User for control operation
CNCExpert	Network Configuration Operator	SYS095148	User for network configuration
CNCAdmin	Administrators	SYS039428	Administrator

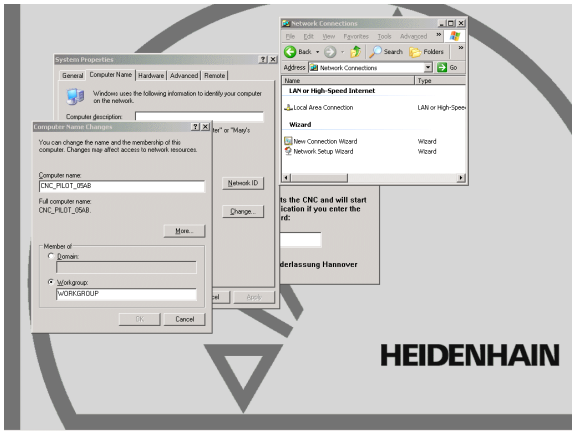
In the factory default setting, the Windows user group "CNCUser" is entered as a workgroup.

You will find general information on the Windows user groups in the Windows online help.

Changing the computer name

Computer name: You must be logged on as an "Administrator" in Windows XP before the computer name can be changed.

- Select "Network Connections > Advanced > Network Identification".
- Enter the new computer name.



Set the workgroup or domain

Selection:

- Select "Setting > Network" in the Transfer mode.

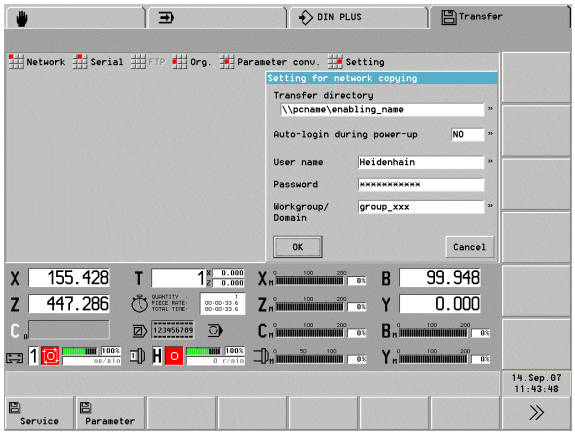
Workgroup: In the "Network Settings" dialog box, set the following parameters that are evaluated when data is exchanged with other computers:

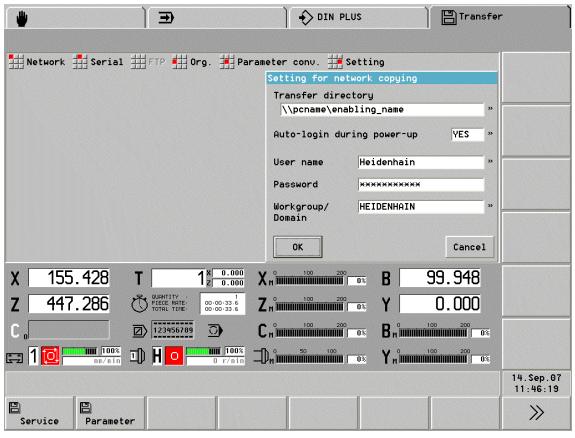
- Transfer directory: Computer name and share name (path) of the directory with which you want to exchange data.
- User name: Name under which the transfer directory is accessed.
- Password: User password
- Workgroup/Domain: Name of the workgroup in which the user name is registered.

Domain: Create an account for the control on the domain controller.

In the "Network Settings" dialog box, set the following parameters that are evaluated when data is exchanged with other computers:

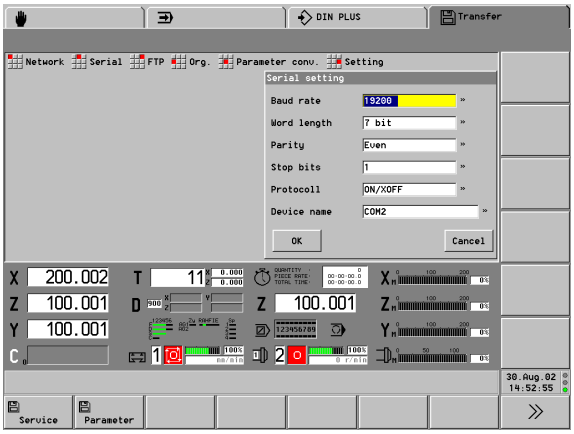
- Transfer directory: Computer name and share name (path) of the directory with which you want to exchange data.
- Auto-login during power-up
  - YES: The control logs onto the given domain with the user name and the password.
  - NO: No automatic logon at startup. Use the logon dialog in Windows.
- User name: Name under which the transfer directory is accessed. The user name is also used for automatic logon at startup.
- Password for network logon
- Workgroup/Domain: Name of the domain





Configuring the serial interface or “printer”

Configure serial interface



- Logon as “System Manager”
- Select “Setting > Serial” in the Transfer mode. The CNC PILOT opens the “Serial setting” dialog box.
- Enter the parameters of the serial interface.

Define the interface parameters in consultation with the communications partner.

- **Baud rate** (in bits per second): The baud rate is defined according to the local condition (cable length, interference, etc.). A high baud rate has the advantage of fast data transfer. A lower baud rate, however, is more stable.
- **Word length:** Choose between 7 or 8 bits per character.
- **Parity:** If you select even/odd parity, the CNC PILOT adds a parity bit so that an even/odd number of set bits are transferred per character. The parity can be checked by the communications partner. If you select “None,” the characters are transmitted they way they have been saved. The parity bit is transmitted in addition to the number of bits defined for the word length.
- **Stop bits:** Choose between 1, 1 1/2 and 2 stop bits.
- **Protocol**
  - Hardware (hardware handshake): The receiver informs the sender through RTS/CTS signals that it is temporarily not able to receive data. A hardware handshake presupposes that the RTS/CTS signals are hardwired in the data transfer cable.
  - XON/XOFF (software handshake): The receiver transmits XOFF if it is temporarily unable to receive data. With XON it signalizes that it can receive more data. The software handshake does not require the transmission of RTS/CTS signals over the data transfer cable.
  - ON/XOFF (software handshake): The receiver transmits XON at the beginning of data transmission to indicate that it is ready to receive. When the receiver is temporarily not able to receive data, it sends XOFF. With XON it signalizes that it can receive more data. The software handshake does not require the transmission of RTS/CTS signals over the data transfer cable.
- **Device name:** COM1/2 identifies the RS-232-C/V.24 data interface

Configure “printer”

- Log in as “System Manager”
- Select “Setting > Printer” in the Transfer mode. The CNC PILOT opens the “Printer setting” dialog box.
- Enter “FILE” in the “Device name” field. The other parameters have no effect.

The printout is prepared and output to a file named “PRINT_xx.txt” (xx: 00 to 19) in the “Data” directory. Maximum file size: 1 MB

For DataPilot, you can also use the “STD” entry for the Windows standard printer.



The parameters of the serial interface are saved in one of the control parameters 41 to 47 (depending on the setting in control parameter 40).

10.2 Data Transfer

Enabling, file types

Shared directories of the CNC PILOT: see table.

Computer systems in the network can access the files located in the shared directories of the CNC PILOT. For security reasons, however, HEIDENHAIN recommends initiating the data exchange from the control.

The network rules of WINDOWS XP apply when accessing shared directories.



Danger of collision!

Other computer systems in the network may overwrite CNC PILOT programs. Organize your network in such a way that only authorized persons have access to the CNC PILOT.

The CNC PILOT distinguishes between the following **file types**, which can be selected in the "Mask of files" dialog box:

- All NC programs: DIN PLUS main programs and subprograms
- NC main programs: DIN PLUS main programs
- NC subprograms: DIN PLUS subprograms
- Expert programs: Special DIN PLUS subprograms
- Template files: DIN PLUS program templates
- Program head lists: Subfiles for program head entries
- Service files: Service files in the DATA directory
- TURN PLUS workpieces: Blank/finished part definitions
- TURN PLUS complete: Blank/finished part definitions and working plans
- TURN PLUS machining sequence: Saved machining sequences
- TURN PLUS blanks: Blank part definitions
- TURN PLUS finished parts: Finished part definitions
- TURN PLUS turret lists: Saved turret assignments
- TURN PLUS contour trains: Descriptions of contour trains
- TURN PLUS DXF files: Contour descriptions in DXF format
- Parameter files: Files of the PARA_USR directory
- Parameter backup: Files of the BACKUP directory

Shared directories of the CNC PILOT	
..\NCPS	NC main programs and subprograms, template files
..\PARA_USR	■ Subfiles for program head entries ■ Converted parameter or operating resource files ■ (Backed-up) error log files
..\DATA	Files for the service personnel
..\BACKUP	Data backup (backup/restore)
TURN PLUS files:	
..\GTR	Descriptions of workpiece blanks
..\GTF	Descriptions of finished parts
..\GTW	Descriptions of workpieces
..\GTC	Complete programs
..\GTT	Descriptions of contour trains
..\GTL	Turret lists
..\GTB	Machining sequences
..\DXF	DXF contours

Basics of operation

Window contents:

- In the left window
  - File transfer: CNC PILOT-specific files
  - Parameters/operating resources: Files in "internal format"
- In the right window
  - File transfer: Files of the communications partner
  - Parameters/operating resources: Files in ASCII format (PARA_USR or BACKUP directory)

Mark files: When using data transfer and organization functions, you mark the file(s) you want to transfer or edit. If no file is marked, the file on which the cursor is currently placed is selected automatically.

Mark	For each file: Place the cursor on the desired file. Press the soft key or + (plus key). The CNC PILOT marks the selected file. Press again to unselect a marked file.

By touchpad: Mark the file by clicking the left or right mouse button.

Click again to unselect a marked file.

Mark all	The CNC PILOT marks all displayed files. Press again to unselect the marked files.

Mask files: The CNC PILOT displays only files that match the selected file type and mask.

Mask	Press the soft key. The CNC PILOT opens the "Mask of files" dialog box.

Set the mask of files:

"File type" field: Press the "Continue" key and select the file type.

"Sort" field: Choose whether you want to sort the files by name or date.

"Mask" field: Enter the mask.

Enter the mask:

- **"*":** Any desired (number of) characters can be specified at this position.
- **"?":** Any desired character can be specified at this position.
- The CNC PILOT
  - automatically adds an asterisk * to the mask you have defined, and
  - displays the current mask setting below the menu line.

Position the cursor

Horizontal arrow keys: Move from left to right window and vice versa. In this way, the CNC PILOT changes between sending and receiving files and between saving and retrieving parameters/ operating resources, respectively.

Vertical arrow keys; **Page Up/Dn**: Move the cursor within the list of files.

Enter character/character string: The cursor positions to the next file name beginning with this character string.

View file (only available for ASCII-format files)

Place the cursor on a DIN PLUS program, parameter file or operating resource file.

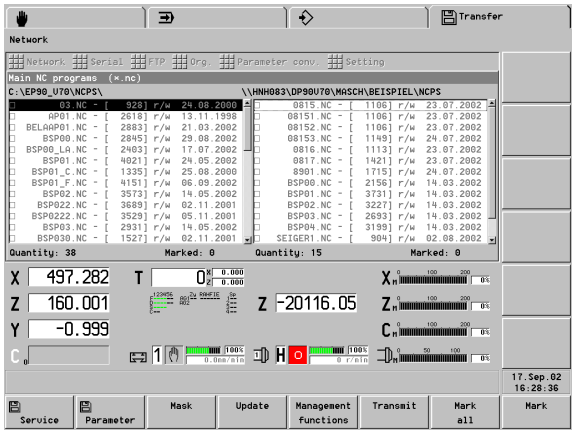
Press ENTER; the CNC PILOT displays the file contents.

Close the file: Press ENTER again (or press the ESC key).

Transmitting and receiving files

- When you select “Network” or “FTP”, the CNC PILOT will display an error message if the communications partner does not respond within a specific time.
- The parameters and operating resource data need to be converted before transfer, and vice versa (see “Parameters and Operating Resources” on page 674).

Data exchange using USB storage media: Enter “D:\” as the transfer directory (dialog box: “Network Settings”). When exchanging data via “Network”, you can access the USB interface at this directory.



Soft keys

Mask	File type, set the mask
Update	Update the file list
Management functions	Call organization functions
Transmit	Transmit the marked files
Receive	■ Ethernet: Get the marked files ■ Serial: Change the CNC PILOT to ready-to-receive state
Mark	Mark a file
Mark all	Mark all files

Ethernet-based transfer

Select “Network” (or “FTP”) in the Transfer menu.

Define a mask to limit the number of files displayed.

Transmit files:

Place the cursor in the left window.

Mark the files you want to transfer.

Transmit — Press the soft key. The CNC PILOT transfers the marked files to the communications partner.

Receive files:

Place the cursor in the right window.

Mark the files you want to load.

Receive — Press the soft key. The CNC PILOT transfers the marked files from the communications partner.

ESC — Press the ESC key to return to the Transfer main menu.

Change communications partners

Log on as NC programmer (or higher).

Select “Setting > Network” (or “FTP”) in the Transfer menu.

Adjust the entry in “Transfer directory” or in “Address/Name of FTP server” for the new communications partner.

Transfer via serial interface

Select “Serial” in the Transfer menu.

The CNC PILOT displays its own files in the left window and the selected interface in the right window.

Define a mask to limit the number of files displayed.

Transmit files:

Mark the files you want to transfer.

Transmit

Press the soft key. The CNC PILOT transfers the marked files over the serial interface.

Receive files:

Receive

Press the soft key. The CNC PILOT changes to ready-to-receive state and receives the transferred data.

ESC

Press the ESC key to return to the Transfer main menu.



During serial transfer, first start the receiver and then the transmitter.

10.3 Parameters and Operating Resources

The CNC PILOT saves parameters and operating resource data in "internal formats." Before data is transferred or backed up, it is converted to ASCII format. Vice versa, i.e. when loading or restoring, the CNC PILOT converts the received parameters/operating resource data back to the "internal format" and integrates it in the active parameter/operating resource files in the control.

When converting to ASCII format, the CNC PILOT saves the data in specific directories. Accordingly, when converting to the "internal format", the CNC PILOT expects the data to be located in the same directories.

With parameters and operating resource data, the CNC PILOT differentiates between:

- **Data exchange (save/load):** Transfer individual files or individual parameters/operating resources. The data is saved/expected in the PARA_USR directory during the conversion.
- **Data backup (backup/restore):** The CNC PILOT backs up **all** parameters/operating resource data or restores all existing backup files. The data is saved/expected in the "BACKUP directory during the conversion.

In the second step, use the standard transfer functions to transmit the files created by the transfer/backup to the destination system. To load parameters/operating resources or restore backed-up files, first transfer them to the appropriate directories of the CNC PILOT before starting the loading or restoring operation.

The CNC PILOT distinguishes between the following **file types** for parameters and operating resources, which can be selected in the "Mask of files" dialog box:

- All: All parameters, operating resources and fixed-word lists
- Tool data: Tool database
- Chucking equipment data: Chucking equipment database
- Fixed-word data: All fixed-word lists
- Technology data: Technology database
- Machine data: Machine parameters
- Control data: Control parameters
- Machining data: Machining parameters
- Setup data: Setup parameters
- PLC data: PLC parameters

Transmitting parameters/operating resources

Place the cursor in the left window.

Transmit a complete file:

Select the parameter group or operating resource group.

Transmit individual parameters/operating resources:

Place the cursor on the parameter group or operating resource group.

Selective collection

Press the soft key. The CNC PILOT lists all the parameters/operating resources of this group.

Mark the parameter/operating resource you want to convert.

Save parameters

Press the soft key. The CNC PILOT opens the “Save parameters” dialog box.

Enter a name for the backup file and select “With comment” or “Without comment”.

The CNC PILOT converts the selected files or the selected individual parameters/operating resources, and saves them in the PARA_USR directory.

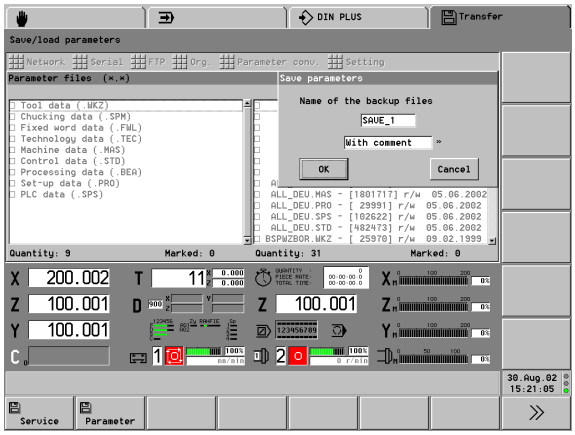
ESC

Press the ESC key to return to the Transfer main menu.

Transfer the created parameter files or operating resource files to the destination system.

Save parameters “with/without comment”:

- Without comment: The transfer function saves only parameters/operating resource data.
- With comment: The transfer function saves the parameters/operating resource data and generates comments for describing the data.



“Send parameters/operating resources” soft key

Selective collection	Select individual parameters/operating resources
Save parameters	Convert the parameters/operating resources to ASCII format
Mark	Mark a file.
Mark all	Mark all files.

Loading parameters/operating resources

The CNC PILOT expects the parameters/operating resource data to be located in the PARA_USR directory.

- The CNC PILOT recognizes the parameter group or operating resource group by the file name extension. This is why the file name can be changed on external systems—but not the extension.
- When reading a file in, the control checks whether the user is authorized to edit the respective parameter and whether automatic mode is active. If the user is not authorized to edit the parameter, the parameter is skipped.

Load parameters/operating resources

Transfer the parameter files or operating resource files to the PARA_USR directory.

Place the cursor in the right window.

Define a mask to limit the number of files displayed.

Mark the files you want to load.

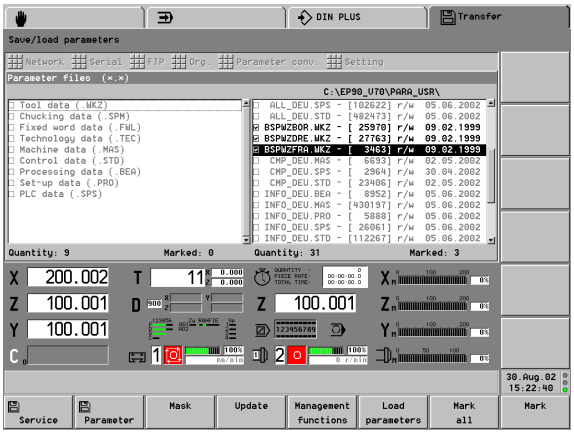
`Load parameters`

Press the soft key. The CNC PILOT converts the data to the “internal format” and integrates them in the control.

As of software version 625 952-05: A confirmation request is generated before the parameters are transferred.

`ESC`

Press the ESC key to return to the Transfer main menu.



“Load parameters/operating resources” soft key

`Mask`	File type, set the mask for the right window
`Update`	Update the file list in the right window
`Management functions`	Call organization functions
`Load parameters`	Convert the parameters/operating resources to the “internal format”
`Mark`	Mark a file.
`Mark all`	Mark all files.

Backing up and restoring data

Create a data backup: Backing up all parameters and operating resources is performed in two steps:

- Select “Backup” to create backup files.
- Use the standard transfer functions to transfer the backup files to an external system.

The **backup** function converts the following data to ASCII format and transfers it to the “BACKUP” directory:

- All parameters
- All operating resource data
- All associated fixed-word lists
- Maintenance system files

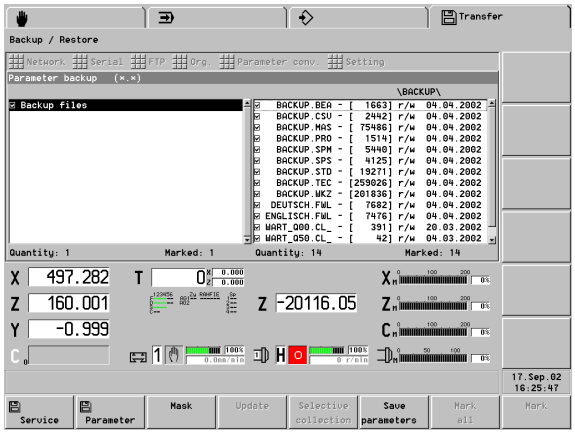
The backup files you created are assigned the file name BACKUP.* and the specific extension of the parameter/operating resource file. The fixed-word lists are assigned the language code as the file name, followed by *.FWL as the file name extension. The backup will overwrite any existing files.

Backup

Select “Parameter conversion > Backup/Restore” in the Transfer menu.

Place the cursor in the left window.

Save parameters	Press the soft key. The CNC PILOT creates the backup files.
ESC	Press the ESC key to return to the Transfer main menu.



“Backup/Restore” soft keys

Mask	Define the sorting order
Update	Update the file list
Save parameters	Start the backup
Load parameters	Start restoring

Restore a data backup: Restoring is performed in two steps:

- Use the standard transfer functions to transfer the backup files from the external system to the BACKUP directory.
- Select "Restore" to convert and integrate the backup files.

Restore downloads all backup files in the BACKUP directory (except maintenance system files).

Restore

Log in as "System Manager"

Select "Parameter conversion > Backup/Restore" in the Transfer menu.

Place the cursor in the right window.

Load parameters

Press the soft key. The CNC PILOT restores the backup files.

ESC

Press the ESC key to return to the Transfer main menu.



- Restore expects a files group saved by backup. HEIDENHAIN recommends that you always treat a file group saved by backup as a block.
- Maintenance-system files can be restored only by trained service personnel.
- Restore cannot be used when automatic mode is active.

Viewing parameter, operating-resource or backup files

Select “Parameter conversion > Save/Load” (or “.. > Backup/ Restore“) in the Transfer menu.

In the right window, place the cursor on a parameter/operating resource file or on a backup file.

Press ENTER; the CNC PILOT displays the file contents.

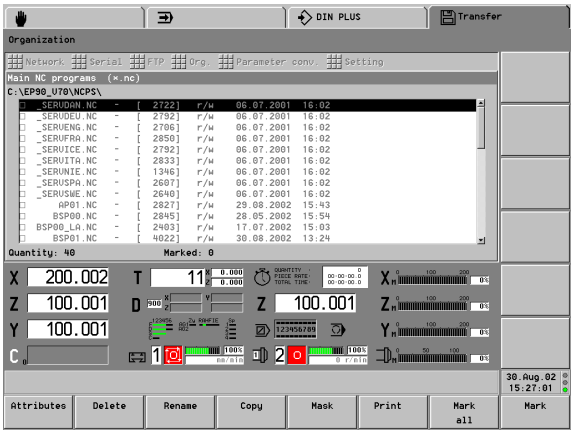
Close the file: Press ENTER again (or press the ESC key).

ESC

Press the ESC key to return to the Transfer main menu.

10.4 File Organization

Fundamentals for file organization



You can organize NC program files and parameter files with the **Copy**, **Delete** and **Rename** functions. The **Print** function is additionally available for ASCII files.

The organization functions are used for CNC PILOT files and, under the following conditions, also for files of the communications partner (external files):

- "WINDOWS Network" data transfer method or USB storage medium
- Log in as "System Manager"

Information in the file list:

- File names and extensions (*.NC = main program; *.NCS = subprogram; etc.)
- File size in bytes (enclosed in [...])
- Attribute
  - r/w: read and write
  - ro: read only
- Date and time of last change
- For NC main programs, the "Drawing" line from the program head is also displayed.

Managing files

Manage CNC PILOT files

Select “Org(anization)” in the Transfer menu.

Define a mask to limit the number of files displayed.

Place the cursor on the parameter or operating resource file.

Mark the required files.

Press ENTER. The CNC PILOT displays the file contents.

Delete	Press the soft key. The CNC PILOT deletes the marked files.
Rename	Press the soft key and enter the new file name. The CNC PILOT renames the file.
Copy	Press the soft key and enter the name of the new file. The CNC PILOT copies the file.
Print	Press the soft key. The CNC PILOT prepares the data for printout and outputs it to the file PRINT_xx.txt (xx: 00 to 19) in the “Data” directory.

Manage CNC PILOT files and external files

Log on as System Manager (or higher).

Select “Network” in the Transfer menu.

Management functions

Press the soft key. The CNC PILOT activates the organization functions for its own files and for those of the communications partner.

Place the cursor in the left or right window.

Place the cursor on the parameter or operating resource file.

Mark the required files.

Press ENTER. The CNC PILOT displays the file contents.

Delete

Press the soft key. The CNC PILOT deletes the marked files.

Rename

Press the soft key and enter the new file name. The CNC PILOT renames the file.

Copy

Press the soft key and enter the name of the new file. The CNC PILOT copies the file.

Print

Press the soft key. The CNC PILOT prepares the data for printout and outputs it to the file PRINT_xx.txt (xx: 00 to 19) in the “Data” directory.



- **Delete:** If no files are marked, the file currently selected with the cursor is deleted.
- **Rename, Copy:** The file currently selected with the cursor is edited.



11

Tables and Overviews

11.1 Undercut and Thread Parameters

Undercut DIN 76, Parameters

TURN PLUS determines the parameters for the thread undercut (undercut DIN 76) from the thread pitch. The undercut parameters are in accordance with DIN 13 for metric threads.

External thread					External thread				
Thread pitch	I	K	R	W	Thread pitch	I	K	R	W
0.2	0.3	0.7	0.1	30°	1.25	2	4.4	0.6	30°
0.25	0.4	0.9	0.12	30°	1.5	2.3	5.2	0.8	30°
0.3	0.5	1.05	0.16	30°	1.75	2.6	6.1	1	30°
0.35	0.6	1.2	0.16	30°	2	3	7	1	30°
0.4	0.7	1.4	0.2	30°	2.5	3.6	8.7	1.2	30°
0.45	0.7	1.6	0.2	30°	3	4.4	10.5	1.6	30°
0.5	0.8	1.75	0.2	30°	3.5	5	12	1.6	30°
0.6	1	2.1	0.4	30°	4	5.7	14	2	30°
0.7	1.1	2.45	0.4	30°	4.5	6.4	16	2	30°
0.75	1.2	2.6	0.4	30°	5	7	17.5	2.5	30°
0.8	1.3	2.8	0.4	30°	5.5	7.7	19	3.2	30°
1	1.6	3.5	0.6	30°	6	8.3	21	3.2	30°

Internal thread					Internal thread				
Thread pitch	**I**	**K**	**R**	**W**	**Thread pitch**	**I**	**K**	**R**	**W**
0.2	0.1	1.2	0.1	30°	1.25	0.5	6.7	0.6	30°
0.25	0.1	1.4	0.12	30°	1.5	0.5	7.8	0.8	30°
0.3	0.1	1.6	0.16	30°	1.75	0.5	9.1	1	30°
0.35	0.2	1.9	0.16	30°	2	0.5	10.3	1	30°
0.4	0.2	2.2	0.2	30°	2.5	0.5	13	1.2	30°
0.45	0.2	2.4	0.2	30°	3	0.5	15.2	1.6	30°
0.5	0.3	2.7	0.2	30°	3.5	0.5	17.7	1.6	30°
0.6	0.3	3.3	0.4	30°	4	0.5	20	2	30°
0.7	0.3	3.8	0.4	30°	4.5	0.5	23	2	30°
0.75	0.3	4	0.4	30°	5	0.5	26	2.5	30°
0.8	0.3	4.2	0.4	30°	5.5	0.5	28	3.2	30°
1	0.5	5.2	0.6	30°	6	0.5	30	3.2	30°

For internal threads, the CNC PILOT calculates the depth of the thread undercut according to the following formula:

Undercut depth = (N + I – K) / 2

Where:

- I: Undercut depth (radius)
- K: Undercut width
- R: Undercut radius
- W: Undercut angle
- N: Nominal diameter of the thread
- I: From the table
- K: Core diameter of the thread

Undercut DIN 509 E, Parameters

Diameter	I	K	R	W
<=1.6	0.1	0.5	0.1	15°
> 1.6 – 3	0.1	1	0.2	15°
> 3 – 10	0.2	2	0. 2	15°
> 10 – 18	0.2	2	0.6	15°
> 18 – 80	0.3	2.5	0.6	15°
> 80	0.4	4	1	15°

The undercut parameters are determined from the cylinder diameter.

Where:

- I: Undercut depth
- K: Undercut width
- R: Undercut radius
- W: Undercut angle

Undercut DIN 509 F, Parameters

Diameter	I	K	R	W	P	A
<=1.6	0.1	0.5	0.1	15°	0.1	8°
> 1.6 – 3	0.1	1	0.2	15°	0.1	8°
> 3 – 10	0.2	2	0.2	15°	0.1	8°
> 10 – 18	0.2	2	0.6	15°	0.1	8°
> 18 – 80	0.3	2.5	0.6	15°	0.2	8°
> 80	0.4	4	1	15°	0.3	8°

The undercut parameters are determined from the cylinder diameter.

Where:

- I: Undercut depth
- K: Undercut width
- R: Undercut radius
- W: Undercut angle
- P: Face depth
- A: Transverse angle

Thread Parameters

To determine the thread parameters, the CNC PILOT uses the following table;

Where:

- F: Thread pitch. Where an asterisk "*" is given in the table, the thread pitch is calculated from the diameter, depending on the thread type (see "Thread pitch" on page 688) .
- P: Thread depth
- R: Thread width
- A: Thread angle, left
- W: Thread angle, right

Calculation formula: Kb = 0.26384*F – 0.1*÷ F

Thread backlash "ac" (depending on thread pitch):

- Thread pitch <= 1: ac = 0.15
- Thread pitch <= 2: ac = 0.25
- Thread pitch <= 6: ac = 0.5
- Thread pitch <= 13: ac = 1

Thread type Q		F	P	R	A	W
Q=1 Metric ISO fine-pitch thread	External	–	0.61343*F	F	30°	30°
	Internal	–	0.54127*F	F	30°	30°
Q=2 Metric ISO thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=3 Metric ISO tapered thread	External	–	0.61343*F	F	30°	30°
Q=4 Metric ISO tapered fine-pitch thread		–	0.61343*F	F	30°	30°
Q=5 Metric ISO trapezoid thread	External	–	0.5*F+ac	0.633*F	15°	15°
	Internal	–	0.5*F+ac	0.633*F	15°	15°
Q=6 Flat metric trapezoid thread	External	–	0.3*F+ac	0.527*F	15°	15°
	Internal	–	0.3*F+ac	0.527*F	15°	15°
Q=7 Metric buttress thread	External	–	0.86777*F	0.73616*F	3°	30°
	Internal	–	0.75*F	F–Kb	30°	3°
Q=8 Cylindrical round thread	External	*	0.5*F	F	15°	15°
	Internal	*	0.5*F	F	15°	15°
Q=9 Cylindrical Whitworth thread	External	*	0.64033*F	F	27.5°	27.5°
	Internal	*	0.64033*F	F	27.5°	27.5°
Q=10 Tapered Whitworth thread	External	*	0.640327*F	F	27.5°	27.5°
Q=11 Whitworth pipe thread	External	*	0.640327*F	F	27.5°	27.5°
	Internal	*	0.640327*F	F	27.5°	27.5°
Q=12 Nonstandard thread		–	–	–	–	–
Q=13 UNC US coarse thread	External	*	0.61343*F	F	30°	30°

Thread type Q		F	P	R	A	W
	Internal	*	0.54127*F	F	30°	30°
Q=14 UNF US fine-pitch thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=15 UNEF U.S. extra-fine-pitch thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=16 NPT U.S. taper pipe thread	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=17 NPTF U.S. taper dryseal pipe thread	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=18 NPSC U.S. cylindrical pipe thread with lubricant	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=19 NPFS U.S. cylindrical pipe thread without lubricant	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°

Thread pitch

Q = 2 Metric ISO thread

Diameter	Thread pitch	Diameter	Thread pitch	Diameter	Thread pitch
1	0.25	6	1	27	3
1.1	0.25	7	1	30	3.5
1.2	0.25	8	1.25	33	3.5
1.4	0.3	9	1.25	36	4
1.6	0.35	10	1.5	39	4
1.8	0.35	11	1.5	42	4.5
2	0.4	12	1.75	45	4.5
2.2	0.45	14	2	48	5
2.5	0.45	16	2	52	5
3	0.5	18	2.5	56	5.5
3.5	0.6	20	2.5	60	5.5
4	0.7	22	2.5	64	6
4.5	0.75	24	3	68	6
5	0.8				

Q = 8 Cylindrical round thread

Diameter	Thread pitch
12	2.54
14	3.175
40	4.233
105	6.35
200	6.35

Q = 9 Cylindrical Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/4“	6.35	1.27	1 1/4“	31.751	3.629
5/16“	7.938	1.411	1 3/8“	34.926	4.233
3/8“	9.525	1.588	1 1/2“	38.101	4.233
7/16“	11.113	1.814	1 5/8“	41.277	5.08
1/2“	12.7	2.117	1 3/4“	44.452	5.08
5/8“	15.876	2.309	1 7/8“	47.627	5.645
3/4“	19.051	2.54	2“	50.802	5.645
7/8“	22.226	2.822	2 1/4“	57.152	6.35
1“	25.401	3.175	2 1/2“	63.502	6.35
1 1/8“	28.576	3.629	2 3/4“	69.853	7.257

Q = 10 Tapered Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7.723	0.907	1 1/2“	47.803	2.309
1/8“	9.728	0.907	2“	59.614	2.309
1/4“	13.157	1.337	2 1/2“	75.184	2.309
3/8“	16.662	1.337	3“	87.884	2.309
1/2“	20.995	1.814	4“	113.03	2.309
3/4“	26.441	1.814	5“	138.43	2.309
1“	33.249	2.309	6“	163.83	2.309
1 1/4“	41.91	2.309			

Q = 11 Whitworth pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/8“	9.728	0.907	2“	59.614	2.309
1/4“	13.157	1.337	2 1/4“	65.71	2.309
3/8“	16.662	1.337	2 1/2“	75.184	2.309
1/2“	20.995	1.814	2 3/4“	81.534	2.309
5/8“	22.911	1.814	3“	87.884	2.309
3/4“	26.441	1.814	3 1/4“	93.98	2.309
7/8“	30.201	1.814	3 1/2“	100.33	2.309
1“	33.249	2.309	3 3/4“	106.68	2.309
1 1/8“	37.897	2.309	4“	113.03	2.309
1 1/4“	41.91	2.309	4 1/2“	125.73	2.309
1 3/8“	44.323	2.309	5“	138.43	2.309
1 1/2“	47.803	2.309	5 1/2“	151.13	2.309
1 3/4“	53.746	1.814	6“	163.83	2.309

Q = 13 UNC U.S. coarse thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0.073“	1.8542	0.396875	7/8“	22.225	2.822222222
0.086“	2.1844	0.453571428	1“	25.4	3.175
0.099“	2.5146	0.529166666	1 1/8“	28.575	3.628571429
0.112“	2.8448	0.635	1 1/4“	31.75	3.628571429
0.125“	3.175	0.635	1 3/8“	34.925	4.233333333
0.138“	3.5052	0.79375	1 1/2“	38.1	4.233333333
0.164“	4.1656	0.79375	1 3/4“	44.45	5.08
0.19“	4.826	1.058333333	2“	50.8	5.644444444
0.216“	5.4864	1.058333333	2 1/4“	57.15	5.644444444
1/4“	6.35	1.27	2 1/2“	63.5	6.35
5/16“	7.9375	1.411111111	2 3/4“	69.85	6.35
3/8“	9.525	1.5875	3“	76.2	6.35
7/16“	11.1125	1.814285714	3 1/4“	82.55	6.35
1/2“	12.7	1.953846154	3 1/2“	88.9	6.35
9/16“	14.2875	2.116666667	3 3/4“	95.25	6.35
5/8“	15.875	2.309090909	4“	101.6	6.35
3/4“	19.05	2.54			

Q = 14 UNF U.S. fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0.06“	1.524	0.3175	3/8“	9.525	1.058333333
0.073“	1.8542	0.352777777	7/16“	11.1125	1.27
0.086“	2.1844	0.396875	1/2“	12.7	1.27
0.099“	2.5146	0.453571428	9/16“	14.2875	1.411111111
0.112“	2.8448	0.529166666	5/8“	15.875	1.411111111
0.125“	3.175	0.577272727	3/4“	19.05	1.5875
0.138“	3.5052	0.635	7/8“	22.225	1.814285714
0.164“	4.1656	0.705555555	1“	25.4	1.814285714
0.19“	4.826	0.79375	1 1/8“	28.575	2.116666667
0.216“	5.4864	0.907142857	1 1/4“	31.75	2.116666667
1/4“	6.35	0.907142857	1 3/8“	34.925	2.116666667
5/16“	7.9375	1.058333333	1 1/2“	38.1	2.116666667

Q = 15 UNEF U.S. extra-fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0.216“	5.4864	0.79375	1 1/16“	26.9875	1.411111111
1/4“	6.35	0.79375	1 1/8“	28.575	1.411111111
5/16“	7.9375	0.79375	1 3/16“	30.1625	1.411111111
3/8“	9.525	0.79375	1 1/4“	31.75	1.411111111
7/16“	11.1125	0.907142857	1 5/16“	33.3375	1.411111111
1/2“	12.7	0.907142857	1 3/8“	34.925	1.411111111
9/16“	14.2875	1.058333333	1 7/16“	36.5125	1.411111111
5/8“	15.875	1.058333333	1 1/2“	38.1	1.411111111
11/16“	17.4625	1.058333333	1 9/16“	39.6875	1.411111111
3/4“	19.05	1.27	1 5/8“	41.275	1.411111111
13/16“	20.6375	1.27	1 11/16“	42.8625	1.411111111
7/8“	22.225	1.27	1 3/4“	44.45	1.5875
15/16“	23.8125	1.27	2“	50.8	1.5875
1“	25.4	1.27			

Q = 16 NPT U.S. taper pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7.938	0.94074074	3 1/2“	101.6	3.175
1/8“	10.287	0.94074074	4“	114.3	3.175
1/4“	13.716	1.411111111	5“	141.3	3.175
3/8“	17.145	1.411111111	6“	168.275	3.175
1/2“	21.336	1.814285714	8“	219.075	3.175
3/4“	26.67	1.814285714	10“	273.05	3.175
1“	33.401	2.208695652	12“	323.85	3.175
1 1/4“	42.164	2.208695652	14“	355.6	3.175
1 1/2“	48.26	2.208695652	16“	406.4	3.175
2“	60.325	2.208695652	18“	457.2	3.175
2 1/2“	73.025	3.175	20“	508	3.175
3“	88.9	3.175	24“	609.6	3.175

Q = 17 NPTF U.S. taper dryseal pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7.938	0.94074074	1“	33.401	2.208695652
1/8“	10.287	0.94074074	1 1/4“	42.164	2.208695652
1/4“	13.716	1.411111111	1 1/2“	48.26	2.208695652
3/8“	17.145	1.411111111	2“	60.325	2.208695652
1/2“	21.336	1.814285714	2 1/2“	73.025	3.175
3/4“	26.67	1.814285714	3“	88.9	3.175

Q = 18 NPSC U.S. cylindrical pipe thread with lubricant

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/8“	10.287	0.94074074	1 1/2“	48.26	2.208695652
1/4“	13.716	1.411111111	2“	60.325	2.208695652
3/8“	17.145	1,411111111	2 1/2“	73.025	3.175
1/2“	21.336	1.814285714	3“	88.9	3.175
3/4“	26.67	1.814285714	3 1/2“	101.6	3.175
1“	33.401	2.208695652	4“	114.3	3.175
1 1/4“	42.164	2.208695652			

Q = 19 NPFS U.S. cylindrical pipe thread without lubricant

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7.938	0.94074074	1/2“	21.336	1.814285714
1/8“	10.287	0.94074074	3/4“	26.67	1.814285714
1/4“	13.716	1.411111111	1“	33.401	2.208695652
3/8“	17.145	1.411111111			

11.2 Pin Layouts and Connecting Cables for the Data Interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices



The interface complies with the requirements of EN 50 178 for "low voltage electrical separation."

Please note that pins 6 and 8 of the connecting cable 274 545 are bridged.

When using the 25-pin adapter block:

CNC PILOT		Connecting cable ID 365 725-xx			Adapter block 310 085-01		Connecting cable ID 274 545-xx		
Pin	**Assignment**	**Socket**	**Color**	**Socket**	**Pin**	**Socket**	**Pin**	**Color**	**Socket**
1	Do not assign	1		1	1	1	1		1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

CNC PILOT		Connecting cable ID 355 484-xx		Adapter block 363 987-02			Connecting cable ID 366 964-xx		
Pin	Assignment	Socket	Color	Pin	Socket	Pin	Socket	Color	Socket
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	White/Green	8	8	8	8	White/Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from the connector assignment of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363 987-02		Connecting cable ID 366 964-xx		
Socket	Pin	Socket	Color	Socket
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

RS-422/V.11 interface

Only non-HEIDENHAIN devices are connected to the RS-422 interface.



- The interface complies with the requirements of EN 50 178 for "low voltage electrical separation."
- The pin layouts on connection X28 (main computer) and on the adapter block are identical.

CNC PILOT		Connecting cable ID 355 484-xx			Adapter block 363 987-02	
Socket	**Assignment**	**Pin**	**Color**	**Socket**	**Pin**	**Socket**
1	RTS	1	Red	1	1	1
2	DTR	2	Yellow	2	2	2
3	RXD	3	White	3	3	3
4	TXD	4	Brown	4	4	4
5	Signal GND	5	Black	5	5	5
6	CTS	6	Violet	6	6	6
7	DSR	7	Gray	7	7	7
8	RXD	8	White/Green	8	8	8
9	TXD	9	Green	9	9	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Not assigned	
5	Not assigned	
6	REC–	Receive Data
7	Not assigned	
8	Not assigned	

11.3 Technical Information

Specifications

CNC PILOT 4290 – Specifications	
Basic version	Contouring control with integrated motor control and integrated inverter ■ 2 closed-loop axes X1 and Z1 on slide 1 ■ 1 feedback-controlled spindle
Expandable to	Maximum 10 control loops ■ Maximum 6 slides ■ Maximum 4 drives ■ Maximum 2 C axes
Components	■ MC 420 or MC 422C main computer ■ CC 422 or CC424 controller unit ■ Operating panel ■ 15-inch TFT color flat-panel display with soft keys
Program memory	Hard disk
Input resolution and display step	■ Linear axes: 0.001 mm ■ B and C axes: 0.001 °
Interpolation	■ **Straight lines:** in 2 principal axes, optionally in 3 principal axes (maximum ±10m) ■ **Circular arcs:** in 2 axes (maximum radius 100 m) ■ **C axis:** interpolation in the linear axes X and Z with the C axis ■ **Helical:** superimposition of circular and straight paths ■ **Look-ahead:** precalculation of up to 20 blocks for determining the contouring velocity profile
Feed rate	■ Input in mm/min or mm/revolution ■ Constant surface speed ■ Feed rate with chip breaking
Data interfaces	■ One each RS-232-C /V.24 and RS-422 / V.11 max. 38.4 kilobaud ■ Ethernet interface 100 Base T (approx. 2 to 5 megabaud, depending on file type and network load) ■ USB 1.1 interface for connecting pointing (mouse) devices and block devices (memory sticks, hard disks, CD-ROM drives)
Surrounding temperature	■ Operation: 0 °C to +45 °C ■ Storage: –30 °C to +70 °C

Accessories

CNC PILOT 4290 – Accessories	
DataPilot	PC software for programming and training for the CNC PILOT 4290 lathe and turning center control: ■ Programming and test runs ■ Program management ■ Operating resource management ■ Backup ■ Training
Electronic handwheel	HR 410 Portable Handwheel

User functions

Standard functions	CNC PILOT 4290
DIN editor	■ Programming in DIN 66025 format
DIN PLUS	■ Setup information on workpiece blank, workpiece material, tools, chucking equipment ■ Extended set of commands (IF...THEN...ELSE; WHILE...; SWITCH...CASE) ■ Dialog-guided user input and help graphics for each programming function ■ Subprograms and variable programming ■ Program verification graphics for workpiece blank and finished part ■ Parallel programming ■ Parallel simulation ■ Alphanumeric program name
Cycles for contour description	■ Standard workpiece blanks ■ Recesses ■ Undercuts ■ Threads ■ Hole patterns on the face and lateral surface (XY and ZY planes) ■ Figure patterns on the face and lateral surface (XY and ZY planes)
Fixed cycles	■ Area clearance cycles, longitudinal and transverse ■ Recessing cycles, radial and axial ■ Recess turning cycles, radial and axial ■ Undercut cycles ■ Parting cycle ■ Thread cycles, radial and axial (multiple threads, successive threads, taper threads, variable pitch) ■ Drilling, deep-hole drilling, and tapping cycles (conventional/rigid), radial and axial (C axis and Y axis) ■ Contour milling and pocket milling, radial and axial (C axis and Y axis) ■ Area milling, centric polygon milling radial and axial (Y axis)

Standard functions	CNC PILOT 4290
TURN PLUS (option 1)	TURN PLUS comprises: ■ Programming with graphic support ■ Graphically supported, interactive sequential programming with DIN PLUS program generation ■ Automatic DIN PLUS program generation with DIN PLUS program generation TURN PLUS is used for: ■ Turning ■ C-axis machining (option 1.1) ■ Y-axis machining ■ Full-surface machining with opposing spindles (option 1.2)
TURN PLUS – Graphical programming	Geometric description of workpieces for the workpiece blank and finished part, including the definition of hole patterns and milling contours for C-axis machining and/or Y-axis machining. Graphical geometry program for calculating and displaying dimensioned or nondimensioned contour points in any length: ■ Simple entry of standardized form elements: chamfers, radii, undercuts, recesses, threads, fits ■ Simple entry of transformations: zero point shift, rotation, mirroring, duplication ■ If more than one geometrical solution exists for calculated coordinates, all of them are presented for selection
C-axis machining (option 1.1)	■ Additional display and programming in the front and surface views (XC/ ZC plane) ■ Hole pattern and figure pattern ■ Creation of freely definable milling contours
Y-axis machining	■ Additional display and programming in the XY and ZY planes ■ Hole pattern and figure pattern ■ Creation of freely definable milling contours
Full-surface machining with opposing spindles (option 1.2)	■ Workpiece description for both setups ■ Description of milling contours and hole patterns also for rear face for C-axis and/or Y-axis machining
DXF import (option)	Loading contours in DXF format: ■ View and select DXF layers ■ Load DXF contour into TURN PLUS

Standard functions	CNC PILOT 4290
TURN PLUS – Sequential programming with interactive graphics	Sequential programming for individual machining steps with: ■ Automatic tool selection ■ Automatic turret assignment ■ Automatic calculation of cutting data ■ Automatic generation of machining sequence in all working planes (also for C-axis machining (with option 1.1) and Y-axis machining) ■ Automatic cutting limitation through chucking equipment ■ Automatic generation of work blocks for rechucking with machine-specific expert program (with option 1.2, full-surface machining) ■ Automatic generation of work blocks for rear-side machining (with option 1.2, full-surface machining) ■ DIN PLUS program generation
TURN PLUS – Automatic sequential programming	Automatic working plan generation with: ■ Automatic DIN PLUS program generation (option) ■ Automatic NC program generation for the rotary axis, C axis, Y axis and for opposing spindles ■ Automatic tool selection ■ Automatic turret assignment ■ Automatic generation of the machining sequence in all working planes ■ Automatic cutting limitation through clamping devices ■ Automatic rechucking with machine-specific expert program for rear-side machining ■ Automatic generation of work blocks for rechucking and for the second setup
Measuring	
In the machine (option 2)	For setting up tools and measuring workpieces in the "Manual" and "Automatic" modes using a touch probe
On external inspection stations (option 3)	Transfer of results from an external measuring device for processing the measured data in the "automatic" operating mode: ■ Up to 16 measuring points ■ RS-232-C/V.24 data interface ■ Communications protocol: 3964-R

SYMBOLS

? – Simplified geometry programming ... 120
/.. Skip level ... 326
variables
In NC program conversion ... 122
Programming ... 316
variables input ... 312
variables output ... 313
$.. Slide code ... 326

SYMBOLE

3-D view ... 382
4-axis machining
Cycle G810 ... 215
Cycle G820 ... 217
9-field box ... 49

A

Absolute coordinates ... 41
Acceleration (slope) G48 ... 192
Access authorization ... 647
Active tool ... 320
Actual value display, display settings ... 571
Actual values in variables G901 ... 306
Adding Tool Dimensions G710 ... 211
Additive compensation
Compensation G149 ... 209
Compensation G149 Geo ... 167
Display ... 98
Address parameters
Fundamentals ... 112
Programming ... 120
Alphabetic keyboard ... 47
Angle offset
C-angle offset G905 ... 287
Measuring angular offset during spindle synchronization G906 ... 288
Angular head ... 613
Area milling
Machining attribute (TURN PLUS) ... 477
Attributes
For overlay elements G39-Geo ... 165
For TURN PLUS contours ... 470
Automatic mode ... 81
Automatic working plan generation (TURN PLUS) ... 534
Auxiliary axes ... 110
Auxiliary contour ... 144
In the simulation ... 364
Auxiliary feed rate ... 497, 642
AWG ... 534
Axis designations ... 40

B

B Axis
Fundamentals ... 34
Backup
General information ... 55
Bar (TURN PLUS) ... 402
Bar grippers ... 613
Basic block display
Automatic mode ... 93
Simulation ... 373
Blank attributes (TURN PLUS) ... 470
Block display
Font size ... 93
Setting ... 93
Block group editing (DIN PLUS) ... 133
Block number
Fundamentals ... 111
Numbering ... 130
Block references
Contour display ... 119
Machining cycles ... 212
Boring G72 ... 248
Button tool ... 612
Byte ... 56

C

C axis
C-angle offset G905 ... 287
Contours machined with ... 118
Fundamentals ... 31
Reference diameter G120 ... 253
Selecting G119 ... 253
Standardizing G153 ... 254
Zero point shift G152 ... 254
Calculating hole positions G840 ... 262
Calculator (TURN PLUS user aid) ... 454
Casting
DIN PLUS casting G21-Geo ... 146
TURN PLUS blank ... 403
Centering
DIN PLUS cycle G72 ... 248
TURN PLUS
Form element ... 415
Front/rear face ... 426
Lateral surface ... 438
Centering tools ... 612
Centric predrilling (IWG) ... 517
Chamfer
DIN cycle G88 ... 238
DIN PLUS cycle G88 ... 238
TURN PLUS form element ... 408
Checking multi-channel programs ... 386
Checking the NC program run ... 383
Chuck part: bar/tube G20-Geo ... 146
Chucking equipment
Database for chucking equipment ... 628
DIN PLUS section code ... 142
Displaying G65 ... 304
Reference points ... 304
Circular arc
DIN PLUS
Front/rear face contour G102-/G103-Geo ... 173
Front/rear face G102, G103 ... 257
Lateral surface contours G112-/G113-Geo ... 180
Lateral surface G112, G113 ... 260
Turning contour G2-, G3-, G12-, G13-Geo ... 148, 150
Turning cycles G2, G3, G12, G13 ... 190, 191
TURN PLUS
Basic contour ... 406
Front/rear face ... 424
Lateral surface ... 437
Circular arc of turning contour G2-, G3-Geo ... 148
Circular interpolation ... 110
Circular paths
Circular path G2/G3 ... 190
Circular pattern with circular slots ... 169
Circular saw blade ... 613

Circular slot
DIN PLUS
Front/rear face G302-/G303-Geo ... 175
Lateral surface G312-/G313-Geo ... 182
In circular patterns ... 169
TURN PLUS
Front/rear face ... 432
Lateral surface ... 444
Comments
Fundamentals ... 112
Input in geometry menu ... 125
Input in machining menu ... 126
Compensation
Additive compensation G149 ... 209
Additive compensation G149-Geo ... 167
Enter the compensation values. ... 87
Compensation of right/left-hand tool tip G150/G151 ... 210
Compensation values ... 622
Configuration
DIN PLUS ... 113
TURN PLUS ... 549
Configuring DIN PLUS
Editing window ... 113
Font size ... 113
Graphic support window ... 113
Connect (TURN PLUS contours) ... 464
Connecting cable for data interfaces ... 694
Constant feed (manual control) ... 64
Constant surface speed Gx96 ... 195
Contour
Activating/deactivating the contour display ... 124
Activating/updating the contour display ... 129
Contour Simulation ... 374
Mirror/shift contour G121 ... 202
Contour definition
DIN PLUS
Basic contour elements ... 147
C-axis contours ... 168
Contour form elements ... 152
Front/rear face ... 172
Fundamentals ... 118
Lateral surface ... 179
Workpiece-blank definition ... 146
TURN PLUS
Contour elements, checking ... 456
Fundamentals for workpiece description ... 396
Workpiece blank contour ... 402
Contour follow-up
Contour follow-up G703 ... 294
Fundamentals ... 119
K default branch G706 ... 295
Saving/loading contour follow-up G702 ... 294
Contour generation during simulation ... 119
Contour machining (finishing) IWG ... 521
Contour milling
DIN PLUS cycle G840 ... 261
TURN PLUS IWG ... 527
TURN PLUS machining attribute ... 476
Contour, description of direction ... 118
Contour-based turning cycles ... 212
Contour-parallel roughing
DIN PLUS
Cycle G830 ... 218
With neutral tool cycle G835 ... 220
TURN PLUS IWG machining ... 502
Control graphics (TURN PLUS) ... 547
Control of the program run ... 330
Controlled parting
By servo-lag monitoring G917 ... 291
By spindle monitoring G991 ... 292
Values for controlled parting G992 ... 293
Controls and displays ... 47
Conventional DIN programming ... 108
Conversion of the NC program ... 122
Converting and mirroring G30 ... 282
Coolant
Technology database ... 642
TURN PLUS IWG ... 497
TURN PLUS machining information ... 553
Coordinates
Coordinate system ... 40
Fundamentals ... 110
Programming ... 120
Coordinates, unknown ... 120
Copy (TURN PLUS)
Circular ... 453
Linear ... 452
Mirroring ... 453
Copying tool ... 612
Counterbore ... 612
Counterboring (TURN PLUS IWG) ... 518
Countersink ... 612
Countersinking
DIN PLUS cycle G72 ... 248
TURN PLUS
Form element ... 415
IWG countersinking ... 518
Cursor ... 56
Cutting data (TURN PLUS IWG) ... 497
Cutting data, determining (TURN PLUS) ... 553
Cutting length ... 622
Cutting limit
Defining/editing (TURN PLUS) ... 484
With preparation (TURN PLUS) ... 482
Cutting material
Designations, specifying ... 649
Technology database ... 641
Cutting path graphics ... 367
Cutting speed
Manual control ... 62
Technology database ... 642
Cycle specification (TURN PLUS IWG) ... 498

D

D display ... 98
D display (display element) ... 97
Data backup
  Transfer mode ... 660
Data exchange (transfer) ... 660
Data input/output (NC program) ... 312
Data interfaces ... 694
Data transfer ... 660
Data transfer methods ... 661
DataPilot ... 660
Date, setting ... 648
Debug ... 374, 376, 380
Deburring
  DIN PLUS cycle G840 ... 268
  TURN PLUS machining attribute ... 478
Deep-hole drilling G74 ... 251
Default value ... 56
Define tool position G712 ... 723
Delete
  Contour elements, deleting (TURN PLUS) ... 462
  Deleting chucking data (TURN PLUS) ... 484
  Deleting machining attributes (TURN PLUS) ... 481
Deleting the chucking data ... 484
Delta drill ... 612
Diagnosis ... 655
Dialog box ... 56
Dialog texts for subprograms ... 328
Digitizing (TURN PLUS user aid) ... 455
Dimensioning (simulation) ... 375
DIN PLUS
  Basic structure ... 108
  Editor ... 124
  Fundamentals ... 30
  Parallel editing ... 109
  Programming ... 108
  Screen ... 109
Direction of contour machining ... 118
Directories ... 668
Display
  Block display ... 93
  DIN PLUS contour display ... 129
  Elements of the machine display ... 97
  Simulation ... 364
  Switch the machine display ... 97
Display of actual values ... 97
Distance-to-go display (display element) ... 97
Drilling
  DIN PLUS
    Cycle for boring, countersinking G72 ... 248
    Cycle for drilling G71 ... 246
    Cycle for tapping G36 ... 250
    Cycle for tapping G73 ... 249
    Deep-hole drilling cycle G74 ... 251
    Fundamentals ... 118
    Hole (centric) G49-Geo ... 162
  TURN PLUS
    Bore hole on lateral surface ... 438
    Centering, countersinking ... 518
    Centric hole ... 415
    Drilling, reaming, deep-hole drilling ... 519
    IWG centric predrilling ... 517
    Machining attributes ... 475
    Tapping ... 520
Drilling cycles
  DIN programming ... 246
Drilling tools ... 612
DXF import ... 460

E

Eccentric polygon
  DIN PLUS
    Eccentric polygon on lateral surface G317 ... 184
    Polygon on front face/rear face G307 ... 177
  TURN PLUS
    Polygon on front or rear face ... 430
    Polygon on lateral surface ... 442
Edit ... 56
Editing switch ... 648
Editing window, setting the (DIN PLUS) ... 113
Element dimensioning (simulation) ... 375
Elements of a DIN program ... 111
Enabling password (network) ... 656
End
  Pocket/island G309-Geo ... 168
  Section code ... 144
End Milling Cutter ... 613
End of cycle G80 ... 231
Engraving
  Character set ... 280
  Engraving on front face G801 ... 279
  Engraving on lateral surface G802 ... 280
Equidistant ... 43
Error log file ... 656
Error message ... 52
Error message (simulation) ... 372
ESC key ... 48
Ethernet
  Data transfer methods ... 661
  Interface RJ45 ... 696
Evaluate events ... 320
Example
  Full-surface machining with opposing spindle ... 346
  Full-surface machining with single spindle ... 349
  Multi-slide operations ... 340, 342
  Programming fixed cycles ... 123
  Subprogram with contour repetitions ... 351
  TURN PLUS ... 562
Expansion options ... 37
Extended input for address parameters ... 120
Extension ... 56
External subprograms ... 327

F

Face roughing G820 ... 215
Feed per minute
  Linear axes G94 ... 194
  Manual control ... 62
  Rotary axes G192 ... 193
Feed per revolution G95 ... 194
Feed rate
  Constant G94 ... 194
  Feed per minute for rotary axes G192 ... 193
  Feed rate override 100% G908 ... 307
  Feed rate reduction factor G38-Geo ... 165
  Feed-rate override, Automatic mode ... 87
  In Manual Control mode ... 62
  Interrupted feed G64 ... 193
  Per revolution G95-Geo ... 166
  Per revolution Gx95 ... 194
  Per tooth Gx93 ... 194
  Rotary axes G192 ... 193
  TURN PLUS attribute ... 472
File management ... 680
File organization ... 680
Files, transmitting/receiving ... 671
Finished part contour
  Fundamentals ... 118
  Section code ... 143
  TURN PLUS ... 397
Finishing
  DIN PLUS
    Cycle G890 ... 228
    Finishing feed rate ... 166
  TURN PLUS
    Clearance turning ... 524
    Contour machining (G890) ... 521
    Undercuts ... 524
Finishing tool ... 612
Fits
  Calculating (TURN PLUS calculator) ... 454
  Finishing – Clearance turning ... 524
  IWG measuring cut ... 524
  TURN PLUS machining information for drilling ... 556
Fixed cycle programming (DIN PLUS) ... 123
Fixed stop, traversing to G916 ... 288
Fixed-word lists ... 649
Font size, setting the (DIN PLUS) ... 113
Forged blank (TURN PLUS) ... 403
Form elements
  DIN PLUS ... 152
  TURN PLUS ... 408
Free editing (DIN PLUS) ... 117
Front face
  Contour definition ... 172
  Machining ... 255
Front window ... 368
Full circle
  DIN PLUS
    Front/rear face G304-Geo ... 176
    Lateral surface G314-Geo ... 183
  TURN PLUS
    Front/rear face ... 428
    Lateral surface ... 440
Full-surface machining
  Fundamentals ... 33
  In DIN PLUS ... 344
  TURN PLUS
    AWG – Machining information ... 560
    AWG – machining sequence ... 536
Functions in Manual mode ... 61

G

G functions
  Manual turning operations ... 64
G functions for contour description
  G0 starting point of turning contour ... 147
  G1 line segment in a contour ... 147
  G10 Surface roughness ... 164
  G100 Starting point of front/rear face contour ... 172
  G101 Line segment in front/rear face contour ... 172
  G102 Circular arc in front/rear face contour ... 173
  G103 Circular arc in front/rear face contour ... 173
  G10-Geo Surface roughness ... 164
  G110 Starting point of lateral surface contour ... 179
  G111 Line segment in a lateral surface contour ... 179
  G112 Circular arc in lateral surface contour ... 180
  G113 Circular arc in lateral surface contour ... 180
  G12 circular arc in a contour ... 150
  G13 circular arc in a contour ... 150
  G149 Additive compensation ... 167
  G2 circular arc in a contour ... 148
  G20 Chuck part: bar/tube ... 146
  G21 casting ... 146
  G22 recess (standard) ... 152
  G23 Recess (general) ... 153
  G24 thread with undercut ... 155
  G25 Undercut contour ... 156
  G3 circular arc in a contour ... 148
  G300 Hole on front face/rear face ... 174
  G301 Linear slot on front/rear face ... 175
  G302 Circular slot on front/rear face ... 175
  G303 Circular slot on front/rear face ... 175
  G304 Full circle on front/rear face ... 176
  G305 Rectangle on front/rear face ... 176
  G307 Eccentric polygon on front/rear face ... 177
  G308 Start pocket/island ... 168
  G309 End of pocket/island ... 168
  G310 Hole on lateral surface ... 181
  G311 Linear slot on lateral surface ... 182
  G312 Circular slot on lateral surface ... 182
  G313 Circular slot on lateral surface ... 182
  G314 Full circle on lateral surface ... 183
  G315 Rectangle on lateral surface ... 183
  G317 Eccentric polygon on lateral surface ... 184
  G34 Thread (standard) ... 159
  G37 Thread (general) ... 160
  G38 Feed rate reduction ... 165
  G39 Attributes for overlay elements ... 165
  G401 Linear pattern on front/rear face ... 177

G402 Circular pattern on front/rear face ... 178
G411 Linear pattern on lateral surface ... 185
G412 Circular pattern on lateral surface ... 186
G49 Hole (centric) ... 162
G52 Blockwise oversize ... 166
G7 Precision stop on ... 164
G8 Precision stop off ... 164
G9 Precision stop blockwise ... 164
G95 Feed per revolution ... 166
G functions for machining
G0 Rapid traverse ... 187
G1 Linear path ... 189
G100 Rapid traverse on front/rear face ... 255
G101 Linear segment on front/rear face ... 256
G102 Circular arc on front/rear face ... 257
G103 Circular arc on front/rear face ... 257
G110 Rapid traverse, lateral surface ... 258
G111 Line segment on lateral surface ... 259
G112 Circular arc on lateral surface ... 260
G113 Circular arc on lateral surface ... 260
G119 No. of C axis ... 253
G12 Circular path ... 191
G120 Reference diameter ... 253
G121 Mirror/shift contour ... 202
G13 Circular path ... 191
G14 Tool change position ... 187
G147 Safety clearance (milling cycles) ... 206
G148 Cutter compensation, switching ... 208
G149 Additive compensation ... 209
G15 Move rotary axis ... 303
G150 Compensation of right-hand tool tip ... 210
G151 Compensation of left-hand tool tip ... 210
G152 Zero point shift, C axis ... 254
G153 Standardize C axis ... 254
G162 Synchronization marking ... 285
G192 Feed per minute for rotary axes ... 193
G2 Circular path ... 190
G204 Wait for moment ... 305
G26 Speed limitation ... 192
G3 Circular path ... 190
G30 Converting and mirroring ... 282
G31 Thread cycle ... 240
G32 Simple thread cycle ... 242
G33 Thread single path ... 244
G36 Tapping ... 250
G4 Period of dwell ... 302
G40 Switch off TRC/MCRC ... 197
G41 Switch on TRC/MCRC ... 197
G42 Switch on TRC/MCRC ... 197
G47 Safety clearance ... 206
G48 Acceleration (slope) ... 192
G50 Switch off oversize ... 204
G51 Zero point shift ... 199
G53 Parameter-dependent zero point shift ... 199
G54 Parameter-dependent zero point shift ... 199
G55 Parameter-dependent zero point shift ... 199
G56 Additive zero point shift ... 200
G57 Axis-parallel oversize ... 204
G58 Contour-parallel oversize ... 205
G59 Absolute zero point shift ... 201
G60 Switch off protection zone ... 303
G600 Preselect tool ... 723
G62 One-sided synchronization ... 284
G63 Synchronous start of slides ... 285
G64 Interrupted feed rate ... 193
G65 Chuck selection ... 304
G66 Component position ... 305
G7 Precision stop on ... 302
G701 Rapid traverse to machine coordinates ... 188
G702 Saving/loading contour follow-up ... 294
G703 Contour follow-up ... 294
G706 K default branch ... 295
G71 Drilling cycle ... 246
G710 Adding tool dimensions ... 211
G717 Updating nominal values ... 305
G718 Move lag error ... 306
G72 Boring, countersinking ... 248
G720 Spindle synchronization ... 286
G73 Tapping ... 249
G74 Deep-hole drilling cycle ... 251
G799 Thread milling, axial ... 278
G8 Precision stop off ... 302
G80 End of cycle ... 231
G801 Engraving on front face ... 279
G802 Engraving on lateral surface ... 280
G81 Simple longitudinal roughing ... 231
G810 Longitudinal roughing ... 212
G82 Simple face roughing ... 232
G820 Face roughing ... 215
G830 Contour-parallel roughing ... 218
G835 Contour-parallel with neutral tool ... 220
G840 Contour milling ... 261
G845 Pocket milling, roughing ... 270
G846 Pocket milling, finishing ... 276
G85 Undercut cycle ... 235
G86 Simple recessing cycle ... 236
G860 Contour-based recessing ... 222
G866 Recessing cycle ... 224
G869 Recess turning cycle ... 225
G87 Line with radius ... 238
G88 Line with chamfer ... 238
G890 Contour finishing ... 228
G9 Precision stop ... 302
G901 Actual values in variables ... 306
G902 Zero point shift in variables ... 306
G903 Lag error in variables ... 306
G905 C-angle offset ... 287
G906 Measuring angular offset during spindle synchronization ... 288
G907 Block speed monitoring off ... 306
G908 Feed rate override 100% ... 307
G909 Interpreter stop ... 307
G910 Switch on in-process measuring ... 296
G912 Actual value determination for in-process measuring ... 296

G913 Switch off in-process measuring ... 296
G914 Switch off probe monitoring ... 296
G915 Post-process measuring ... 298
G916 Traversing to a fixed stop ... 288
G917 Controlled parting ... 291
G918 Velocity feedforward ... 307
G919 Spindle override 100% ... 307
G920 Deactivating zero shifts ... 308
G921 Deactivating zero-point shifts, tool lengths ... 308
G922 shaft speed with V constant ... 311
G93 Feed per tooth ... 194
G930 Sleeve monitoring ... 310
G933 Thread switch ... 239
G94 Constant feed rate ... 194
G940 T no. internal ... 308
G941 Transferring magazine compensation values ... 309
G95 Feed per revolution ... 194
G96 Constant surface speed ... 195
G97 Spindle speed ... 195
G975 Servo lag limit ... 309
G98 Spindle with workpiece ... 283
G980 Activating zero point shifts ... 309
G981 Activating zero point shifts, tool lengths ... 310
G99 Workpiece group ... 284
G991 Controlled parting – spindle monitoring ... 292
G992 Values for controlled parting ... 293
G995 Specifying the monitoring zone ... 301
G996 Type of monitoring ... 301
Simple contour repeat cycle G83 ... 234
G machining commands
G1 Linear path (milling) ... 723
G16 Tilting the working plane ... 723
G17 XY plane (front or rear face) ... 723
G18 XZ plane (turning) ... 723
G701 Rapid traverse to machine coordinates ... 723
G712 Define tool position ... 723
Global variables (DIN programming) ... 316
Graphic display ... 94
Graphic windows ... 129
Graphic, magnifying/reducing
Simulation ... 371
TURN PLUS ... 547
Graphics (DIN PLUS) ... 124
Gripping device ... 613
Guarding ring (TURN PLUS) ... 413

H

Handwheel ... 47, 65
Help ... 50
Help commands for contour definition ... 163
Help graphic, setting the (DIN PLUS) ... 113
Help graphics for subprogram calls ... 329
Hole
DIN PLUS
Hole on front face/rear face G300 ... 174
Hole on lateral surface G310 ... 181
TURN PLUS
Single hole on front or rear face ... 426
Hollowing
TURN PLUS IWG
Residual roughing – contour-parallel ... 505
Residual-contour machining ... 521
Roughing ... (neutral tool) ... 506
TURN PLUS machining information ... 554

I

Identification number
Chucking equipment ... 142
Tool ... 137
IF.. Program branching ... 322
inch
Machine mode of operation 61, 81
Inches
Programming ... 111
Incremental address parameters
Identifier ... 112
Programming ... 120
Incremental coordinates ... 41
Indexable-insert drill ... 612
Info system ... 50
In-process measuring ... 296
INPUT (input of # variable) ... 312
Input and output
Operator communication ... 112
Programming ... 312
Time for ... 122
Input box ... 48
Input field ... 48
INPUTA (input of V variables) ... 314
INS key 48, 49
Inserting (TURN PLUS contour) ... 463
Inside contours: TURN PLUS machining information ... 555
Inspection mode ... 89
Inspector (TURN PLUS user aid) ... 456
Instructions (DIN PLUS) ... 126
Integer variable ... 315
Interactive Working Plan Generation (IWG) ... 494
Interface specifications ... 694
Internal error ... 53
Interpreter stop G909 ... 307
Interrupted feed G64 ... 193
Inverting, TURN PLUS transformations ... 469
Island (DIN PLUS) ... 168
Isolating a detail
Simulation ... 371
TURN PLUS ... 547
IWG ... 494

J

Jog keys ... 65

L

L call ... 327
Lag error (following error)
Ignoring G718 ... 306
In variables G903 ... 306
Limit G975 ... 309
Language, setting ... 648
Lateral surface
Contour commands ... 179
Machining commands ... 258
Reference diameter G120 ... 253
Lathe tools ... 612
Left-hand or right-hand tool ... 622

Line segment
DIN PLUS
Front/rear face contour G101-Geo ... 172
Front/rear face G101 ... 256
Lateral surface contour G111-Geo ... 179
Lateral surface G111 ... 259
Linear path G1 ... 189
Turning contour G1-Geo ... 147
With chamfer G88 ... 238
With radius G87 ... 238
TURN PLUS
Lateral surface ... 436
Turning contour ... 405
Linear and rotary axes ... 110
Linear path G1 ... 189
Linear path G1 (milling) ... 723
Linear path G101 ... 256
Linear path G111 ... 259
Load display (display element) ... 97
Load monitoring
Analyzing reference machining ... 104
Editing limit values ... 103
Fundamentals ... 100
Monitoring zone definition G995 ... 301
Parameters for ... 105
Production under ... 103
Programming ... 300
Reference machining ... 102
Type of monitoring G996 ... 301
Working with ... 101
Local variables (DIN programming) ... 316
Log file ... 656
Longitudinal roughing G810 ... 212
Longitudinal roughing, simple G81 ... 231

M

M ... 331
M commands
In Manual Control mode ... 63
M97 Synchronous function ... 286
M99 program end with return jump ... 330
TURN PLUS program head ... 393
M commands in DIN PLUS programming ... 330
M commands, TURN PLUS ... 497
Machine commands ... 331
Machine Data ... 62
Machine datum ... 40
Machine display
Adjusting/switching ... 97
Display elements ... 97
Fundamentals ... 46
Parameters for machine display ... 580
Machine operating panel ... 47
Machine parameters (MP) ... 571
Machine reference points ... 40
Machining information (TURN PLUS) ... 552
Machining modes (TURN PLUS IWG)
Drilling ... 516
Finishing ... 521
Milling ... 526
Recessing ... 507
Roughing ... 499
Thread ... 525
Machining sequence AWG
Editing ... 536
General ... 535
List of machining sequences ... 538
Managing ... 536
Machining simulation ... 376
Machining with DIN PLUS
Machining menu ... 126
Section code ... 144
Main feed rate ... 497
Maintenance system ... 650
Manual direction keys ... 65
Material (technology database) ... 641
Material designations ... 649
Mathematical functions ... 315
Measure
TURN PLUS machining attribute ... 473
Measuring
In-process measuring ... 296
Measuring tools with a probe ... 79
Measuring tools with an optical gauge ... 79
Post-process measuring ... 298
Tool measurement ... 79
Tool measurement by touching the workpiece ... 79
Measuring systems ... 613
Menu items ... 49
Metric
Dimensional system in Automatic mode ... 81
Dimensional system in Manual control mode ... 61
Overview of units of measure ... 42
Mid-program startup ... 84
Milling
DIN PLUS
Contour milling G840 ... 261
Fundamentals ... 118
Pocket milling, finishing G846 ... 276
Pocket milling, roughing G845 ... 270
TURN PLUS
Area milling attribute ... 477
Contour milling attribute ... 476
IWG milling ... 526
Milling cutter radius compensation
Fundamentals ... 43
Programming ... 196
Milling cycles
DIN PLUS
Contour milling G840 ... 261
Engraving on front face G801 ... 279
Engraving on lateral surface G802 ... 280
Engraving, character set ... 280
Pocket milling, finishing G846 ... 276
Pocket milling, roughing G845 ... 270
TURN PLUS
Area milling ... 531
Contour milling ... 527
Deburring ... 529
Engraving ... 530
Milling direction (DIN PLUS)
Cycle G840 ... 263
Cycle G845 ... 270
Cycle G846 ... 276
Milling pins ... 613
Milling tools ... 613

Mirroring
DIN PLUS
Converting and mirroring G30 ... 282
Mirror/shift contour G121 ... 202
TURN PLUS
Copying a contour section by mirroring ... 453
Transformations – Mirroring ... 469
Modal address parameters ... 120
Modal G functions ... 120
Monitoring zone definition G995 ... 301
Motion simulation ... 380
Mount type ... 622
Multipoint tools
Tool parameters ... 620
Tool programming ... 121
Multi-slide programming
Example of dual-slide machining ... 338, 340
Example of positioning the steady rest ... 334
Example with four-axis cycle ... 342
Example with traveling steady rest ... 336
Overview ... 332
Program run ... 334

N

Navigate ... 56
NC address parameters ... 112
NC block number increment ... 130
NC blocks
Fundamentals ... 111
Making ... 115
Numbering ... 130
NC center drill ... 612
NC commands
Editing, deleting ... 115
Fundamentals ... 111
NC elements
Changing ... 115, 116
Deleting ... 115
NC program conversion ... 122
NC program header ... 124
NC program sections ... 108
NC subprograms ... 122
Negative X coordinates ... 110
Nested contours ... 168
Networks
Configuration ... 663
Overview ... 661
Settings (diagnosis) ... 656

O

OK button 49
One-sided synchronization G62 ... 284
Open contours ... 118
Operating modes
Automatic mode ... 81
DIN PLUS ... 108
Manual control ... 61
Overview ... 35
Parameters ... 568
Selecting the operating mode ... 48
Service and Diagnosis ... 646
Simulation ... 362
Transfer ... 660
TURN PLUS ... 390
Operation
Data input ... 48
Function selection ... 48
List operations ... 48
Menu selection ... 48
Screen buttons ... 48
Selecting the operating mode ... 48
Soft keys ... 48
Operator communication ... 112
Options ... 37
Options, overview of ... 656
Organization (file management) ... 680
Outputs
variables ... 313
Operator communication ... 112
Programming ... 312
Time for ... 122
V variable ... 314
Overhang length ... 622
Overlay element (TURN PLUS)
Circular arc ... 418
Integrating overlay elements ... 399
Pontoon ... 419
Wedge ... 418
Override knob ... 47
Oversize
Axis-parallel G57 ... 204
Blockwise G95-Geo ... 166
Contour parallel (equidistant) G58 ... 205
Switching off G50 ... 204
TURN PLUS attribute ... 471
Overview of enabled elements (display element) ... 97
Overview of enabled elements (machine display) ... 97

P

Parallel editing (DIN PLUS) ... 114
Parallel operation ... 108
Parameter
Loading parameters/operating resources ... 676
Parameters/operating resources - creating a data backup ... 677
Parameters/operating resources - restoring a data backup ... 677
Transferring parameters and operating resources ... 674
Transmitting parameters/operating resources ... 675
Viewing parameter, operating-resource or backup files ... 679
Parameter description – subprograms ... 328
Parameter-dependent zero point shift G53 ... G55 ... 199
Parameters
Control parameters for machine display ... 580
Control parameters for simulation ... 579
Editing ... 569
Editing Configuration Parameters ... 570
General control parameters ... 577
General machine parameters ... 571
Machine parameters for C axes ... 574
Machine parameters for linear axes ... 575
Machine parameters for slides ... 572
Machine parameters for spindles ... 573
Machining parameters ... 585
Parameter values, reading (DIN PLUS) ... 316
Set-up parameters ... 583
Part handling systems ... 613
Parting (IWG)
Standard machining ... 512
Parting tool ... 612
Password ... 647

Pattern
DIN PLUS
Circular front/rear face G402-Geo ... 178
Circular lateral surface G412-Geo ... 186
Linear front/rear face G401-Geo ... 177
Linear lateral surface G411-Geo ... 185
TURN PLUS
Circular, front/rear face ... 434
Circular, lateral surface ... 446
Linear, front/rear face ... 433
Linear, lateral surface ... 445
Period of dwell G4 ... 302
Picture number of tool graphic ... 622
Pin layout for data interfaces ... 694
PLC message 54
Pocket milling
DIN PLUS
End of pocket G309 ... 168
Pocket milling, finishing G846 ... 276
Pocket milling, roughing G845 ... 270
Start pocket G308 ... 168
TURN PLUS
IWG pocket milling – Roughing/Finishing ... 531
Milling depth ... 421
Point dimensioning ... 375
Polar coordinates ... 41
Polygon
DIN PLUS
Front/rear face G307-Geo ... 177
Lateral surface G317-Geo ... 184
TURN PLUS
Front/rear face ... 430
Lateral surface ... 442
Position angle ... 622
Position display ... 97
Position display (display element) ... 97
Position encoders ... 39
Position nominal values, updating G717 ... 305
Position of milling contours
DIN PLUS ... 168
TURN PLUS front/rear face ... 421
TURN PLUS lateral surface ... 421
Post-process measuring
Cycle G915 ... 298
Status ... 96
Precision stop
DIN PLUS attribute for contour description ... 164
DIN PLUS machining commands ... 302
TURN PLUS attribute ... 479
Predrilling (IWG) ... 517
Preparing a machining process (TURN PLUS) ... 482
Chucking a workpiece at the spindle ... 483
Chucking a workpiece at the tailstock ... 483
Defining a cutting limitation ... 484
Deleting the chucking data ... 484
Fundamentals ... 482
Rechuck – 1st setup after 2nd setup ... 486
Rechuck – Standard machining ... 485
Setting up a tool list ... 490
Preselect tool G600 ... 723
Primary cutting edge ... 121
Principal axes
Arrangement ... 40
Fundamentals ... 110
PRINT (output of # variable) ... 313
PRINTA (output of V variables) ... 314
Program branching
Fundamentals ... 112
Programming ... 322
Program branching, IF ... 322
Program branching, SWITCH ... 324
Program branching, WHILE ... 323
Program conversion ... 122
Program end with restart ... 330
Program head
DIN PLUS ... 136
TURN PLUS ... 393
Program number ... 111
Program run, modifying ... 85
Program section codes ... 135
Program selection ... 82
Programming with variables ... 315
Protection zone
Define ... 77
Protection zone and limit switch monitoring (machining simulation) ... 377
Protection zone and limit switch monitoring (motion simulation) ... 381
Switching off G60 ... 303
Pull-down menu ... 48

Q

Quantity of workpieces/time per workpiece (display element) ... 97
Quantity, monitoring for number of parts produced
Defining quantity ... 85
Quantity of workpieces/time per workpiece (display element) ... 97
Tool life management ... 72
Tool life monitoring in the tool database ... 620

R

Radius G87 ... 238
Rapid paths (simulation) ... 367
Rapid traverse
Front/rear face G100 ... 255
In machine coordinates G701 ... 188
Lateral surface G110 ... 258
Rapid traverse G0 ... 187
Rapid traverse to machine coordinates G701 ... 723
Real variables ... 315
Reamer ... 612
Rear-face machining
DIN PLUS
Elements of the front/rear face contour ... 172
Example of full-surface machining with opposing spindle ... 346
Example of full-surface machining with single spindle ... 349
Section code ... 144

Rear-side machining
DIN PLUS
Section code ... 144
TURN PLUS
Machining sequence ... 536
Preconditions for full-surface machining ... 560
Recess turning
DIN PLUS cycle G869 ... 225
TURN PLUS IWG recess turning radial/axial ... 510
Recessing
DIN PLUS
Contour based recessing G860 ... 222
Recess contour (general) G23-Geo ... 153
Recess contour (standard) G22-Geo ... 152
Recessing cycle G866 ... 224
Recessing G860 ... 222
Simple G86 ... 236
Simple G866 ... 224
TURN PLUS
Form element, general recess ... 412
Form element, recess type D (sealing ring) ... 412
Form element, recess type F (relief turn) ... 413
Form element, recess type S (guarding ring) ... 413
IWG contour recessing radial/axial ... 508
IWG recessing radial/axial ... 509
Recessing (IWG) ... 509
Recessing tool ... 612
Recess-turning tool ... 612
Rectangle
DIN PLUS
Front/rear face G305-Geo ... 176
Lateral surface G315-Geo ... 183
TURN PLUS
Front or rear face ... 429
Lateral surface ... 441
Reference data, TURN PLUS
Front and rear face ... 421
Lateral surface ... 421
Reference diameter
Reference diameter G120 ... 253
Reference diameter G308 ... 168
Reference mark ... 39
Reference plane
FRONT section ... 144
REAR SIDE section ... 144
Reference plane G308 ... 168
SURFACE section ... 144
Reference point setting/canceling (simulation) ... 375
Relief turn
Form element G23-Geo ... 153
TURN PLUS form element ... 413
Remote diagnosis ... 656
Replacement tool ... 121
Replacement tools
Defining the replacement sequence ... 72
Fundamentals ... 121
Tool life management ... 72
Residual-contour machining
DIN PLUS residual-contour machining ... 230
TURN PLUS
IWG finishing ... 521
IWG residual roughing – contour-parallel ... 505
IWG residual roughing – longitudinal ... 503
IWG residual roughing – transverse ... 504
Resolve (TURN PLUS) ... 464
Restarting the NC program ... 82
RETURN [BLANK] (section code) ... 145
ROHTEIL [BLANK] (section code) ... 143
Rotary axis
Feed per minute for rotary axes G192 ... 193
Fundamentals ... 110
Moving G15 ... 303
Roughing
DIN PLUS
Contour-parallel roughing G830 ... 218
Contour-parallel with neutral tool G835 ... 220
Face roughing G820 ... 215
Longitudinal roughing G810 ... 212
TURN PLUS
Contour-parallel ... 502
Hollowing (neutral tool) ... 506
IWG residual roughing – contour-parallel ... 505
IWG residual roughing – longitudinal ... 503
IWG residual roughing – transverse ... 504
IWG roughing longitudinal ... 500
IWG roughing transverse ... 501
Roughing tool ... 612
Rounding
TURN PLUS form element ... 408
Rounding arc
DIN PLUS cycle G87 ... 238
Run-out length (thread) ... 239

S

Safety clearance
Milling G147 ... 206
Turning G47 ... 206
Screen ... 47
Screen buttons ... 48
Screen displays
DIN PLUS screen ... 110
General ... 46
Simulation screen ... 363
Sealing ring (TURN PLUS form element) ... 412
Secondary axes ... 110
Secondary machining direction ... 622
Section codes, overview ... 135
Sectional view (simulation) ... 374
Selectable stop
Automatic mode ... 86
M01 ... 330
Semiautomatic (IWG) ... 494
Separation point
TURN PLUS attribute ... 480
TURN PLUS machining information ... 557

Sequential events ... 320
Serial interface ... 666
Service functions ... 647
Setting dimensions ... 622
Setting the feed per revolution ... 62
Setting up machine dimensions ... 78
Setting up the chucking table ... 74
Setup
  DIN PLUS program head ... 136
  Setup functions ... 75
  TURN PLUS program head ... 393
Set-up parameters ... 583
Shaft machining (TURN PLUS)
  Fundamentals ... 557
  Preparing a machining process ... 482
Shaft speed
  Spindle speed override ... 87
Shaft speed with V constant G922 ... 311
Sharing
  Enabling password ... 656
  Shared directories ... 668
Shift, activating tool lengths G981 ... 310
Shifting the contour G121 ... 202
Side mill cutter ... 613
Side view (YZ) (simulation) ... 368
Simple contour repeat cycle G83 ... 234
Simple face roughing G82 ... 232
Simple tools
  Programming ... 140
  Setup ... 71
Simplified geometry programming ... 120
Simulation
  3-D view ... 382
  Checking multi-channel programs ... 386
  Chucking equipment depiction ... 364
  Contour generation during simulation ... 378
  Contour Simulation ... 374
  Dimensioning ... 375
  Display ... 364
  Errors and warnings ... 372
  Front window ... 368
  Line and trace display ... 367
  Machining simulation ... 376
  Motion simulation ... 380
  Operating mode 362
  Screen contents ... 363
  Side view (YZ) ... 368
  Surface window ... 368
  Synchronous point analysis ... 388
  Time calculation ... 387
  Tool depiction ... 364
  TURN PLUS control graphics ... 548
  Zero point shifts ... 366
  Zoom function ... 371
Single hole (TURN PLUS) ... 426
Single-block mode
  Automatic operating mode ... 86
  Simulation ... 363
Skip cycle ... 326
Skip level
  Execution ... 326
  Fundamentals ... 112
Sleeve monitoring G930 ... 310
Slide change key ... 66
Slide code
  Conditional block run ... 326
  Fundamentals ... 112
Slide display (display element) ... 97
Slide synchronization ... 282
  General ... 282
  Multi-slide programming ... 332
  One-sided synchronization G62 ... 284
  Synchronization marking G162 ... 285
  Synchronous start of slides G63 ... 285
Slot
  DIN PLUS
    Circular slot on front/rear face G302/G303 ... 175
    Circular slot on lateral surface G312/G313 ... 182
    Linear slot on front/rear face G301 ... 175
    Linear slot on lateral surface G311 ... 182
  TURN PLUS
    Circular slot on front/rear face ... 432
    Circular slot on lateral surface ... 444
    Linear slot on front/rear face ... 431
    Linear slot on lateral surface ... 443
Slot, linear
  DIN PLUS
    Front/rear face G301-Geo ... 175
    Lateral surface G311-Geo ... 182
  TURN PLUS
    Front/rear face ... 431
    Lateral surface ... 443
Software limit switch
  Manual control ... 61
  Reference run ... 58
Source block display (simulation) ... 369
Special machining tasks (SM) ... 532
Spindle
  Spindle change key ... 66
  Spindle condition ... 99
  Spindle keys ... 65
  Spindle override 100% G919 ... 307
  Spindle speed ... 62
  Spindle synchronization G720 ... 286
  With workpiece G98 ... 283
Spindle display (display element) ... 97
Spindle keys ... 65
Spindle point stop ... 62, 63
Spindle speed
  Block speed monitoring off G907 ... 306
  Constant surface speed Gx96 ... 195
  Speed Gx97 ... 195
  Speed limitation Gx26 ... 192

Start pocket/island G308-Geo ... 168
Starting length (thread) ... 239
Starting point of contour
DIN PLUS
Display ... 129
Front/rear face G100-Geo ... 172
Lateral surface G110-Geo ... 179
Turning contour G0-Geo ... 147
TURN PLUS
Basic contour ... 404
Front/rear face ... 422
Lateral surface ... 435
Starting template ... 354
Step drill ... 612
Stopper tool ... 613
Structure template ... 354
Structured DIN PLUS program ... 108
Subprogram
Call ... 327
Fundamentals ... 122
Section code ... 145
Surface roughness
Machining parameters ... 585
Surface roughness G10 ... 164
TURN PLUS attribute ... 472
Surface window ... 368
SWITCH..CASE—program branching ... 324
Switch-off ... 60
Switch-on ... 58
Synchronization
One-sided G62 ... 284
Synchronization marking G162 ... 285
Synchronization, spindle G720 ... 286
Synchronous function M97 ... 286
Synchronous start of slides G63 ... 285
Synchronous point analysis ... 388
System error ... 53

T

T commands
Fundamentals ... 121
Tool change ... 207
T display (display element) ... 97
Tables
Thread pitch ... 688
Undercut DIN 509 E, Parameters ... 686
Undercut DIN 509 F, Parameters ... 686
Undercut DIN 76, Parameters ... 684
Tapping
DIN PLUS
Cycle G36 ... 250
Thread, contour-based G73 ... 249
TURN PLUS
Centric hole ... 415
Front/rear face ... 426
IWG tapping ... 520
Lateral surface ... 438
Taps ... 612
Technical information ... 697
Technology database ... 641
Template control ... 354
Thread
DIN PLUS
Single path G33 ... 244
Tapping G36 ... 250
Thread cycle G31 ... 240
Thread cycle, simple G32 ... 242
Thread switch G933 ... 239
TURN PLUS
Form element ... 414
IWG machining ... 525
Machining attributes ... 474
Thread milling cutter ... 613
Thread milling, axial G799 ... 278
Thread overrun ... 239
Thread parameters ... 687
Thread pitch ... 688
Threading tool ... 612
Threads
DIN PLUS
General G37-Geo ... 160
Standard G34-Geo ... 159
With undercut G24-Geo ... 155
Tilted position of tool carrier ... 121
Tilted working plane - Basics ... 34
Tilting the working plane G16 ... 723
Time calculation ... 387
Time, setting ... 648
Tool
Displaying the tool graphic ... 611
Inserting (DIN PLUS) ... 207
Measurement ... 79
Tool database ... 608
Tool display (simulation) ... 364
Tool call (TURN PLUS IWG) ... 497
Tool change position
Setting the tool changing point ... 75
Tool changing point G14 ... 187
Tool compensation
Programming with variables ... 318
Tool compensation in the Automatic mode ... 87
Tool compensation, calculating ... 80
Tool dimensions ... 43
Tool dimensions, adding G710 ... 211
Tool edge compensation G148 ... 208
Tool edge compensation, switching G148 ... 208
Tool ID number ... 622
Tool life management
Data entry ... 72
Data in the tool database ... 620
In Automatic mode ... 88
Parameter entry ... 72
Tool diagnosis bits ... 318
Tool life monitoring
Tool life monitoring in the tool database ... 620
With load monitoring ... 300
Tool list
Compare with NC program ... 70
Setup (machine setup) ... 68
Setup (TURN PLUS) ... 490
Transferring from NC program ... 71
Tool positioning without machining ... 187
Tool programming ... 121
Tool selection
Tool changing, manual control ... 63
TURN PLUS ... 552
Tool type ... 622
Tool types, overview ... 612
Tool-tip radius compensation
Fundamentals ... 43
Programming ... 196

Touchpad ... 47
Transfer ... 660
Transformations (TURN PLUS contours) ... 467
Tube (TURN PLUS) ... 402
TURN PLUS ... 30
AWG
Editing and managing machining sequences ... 536
List of machining sequences ... 538
Machining sequence ... 535
Contour definition
Assigning attributes ... 470
Attributes for workpiece blanks ... 470
Colors of selection points ... 448
Connect ... 464
Contours of workpiece blanks ... 402
Editing the contours of a blank part ... 461
Entering C-axis contours ... 400
Entering the finished part contour ... 397
Entering the workpiece blank contour ... 396
Help functions for element definition ... 447
Inserting a contour ... 463
Integrating a contour train ... 399
Overlay elements ... 418
Resolving (form elements, figures, patterns) ... 464
Superimposing form elements ... 398
Transformations ... 467
Workpiece description ... 396
General
Configuration ... 549
Control graphics ... 547
Example ... 562
Machining information ... 552
Managing files ... 391
Operating mode ... 390
Program head ... 393
IWG
Cutting data ... 497
Cycle specification ... 498
Finishing ... 521
Interactive working plan generation ... 494
Milling ... 526
Special machining tasks ... 532
Thread machining ... 525
Tool call ... 497
Machining information
Cutting parameters ... 553
Drilling ... 556
Full-surface machining ... 560
Hollowing ... 554
Inside contours ... 555
Shaft machining ... 557
Tool selection ... 552
Turret assignment ... 552
Preparing a machining process
Defining a cutting limitation ... 484
Setting up a tool list ... 490
Turning contours ... 118
Turning cycles
Contour-based ... 212
Simple ... 231
Turning direction (direction of rotation) ... 622
Turn-out tool ... 612
Turret
DIN PLUS tool programming ... 121
Section code, TURRET ... 137
TURN PLUS turret assignment ... 552
Twist drill ... 612
Twist drill cutter ... 613
Type of monitoring G996 ... 301

U

Undercut
DIN PLUS
Cycle G85 ... 235
Definition with G25-Geo ... 156
DIN 509 E ... 157
DIN 509 F ... 157
DIN 76 ... 158
Type H ... 158
Type K ... 159
Type U ... 156
TURN PLUS
Type E ... 409
Type F ... 409
Type G (DIN 76) ... 409
Type H ... 410
Type K ... 410
Type U ... 410
Undercut parameters
DIN 509 E ... 686
DIN 509 F ... 686
DIN 76 ... 684
Units of measure
Overview ... 42
Units of measurement
In the DIN PLUS program ... 111
Units, specifying ... 136
Update nominal values G717 ... 305
USB storage media ... 661
User aid (TURN PLUS)
Calculator ... 454
Contour elements, checking ... 456
Digitizing ... 455
Error messages ... 457
Selections ... 448
Unresolved contour elements ... 447
Zero point shift ... 452
User, entering ... 647

V

Values for controlled parting G992 ... 293
Variables
- # variables ... 316
- As address parameters ... 120
- Assignment ... 320
- Calculations ... 315
- Scope (# variables) ... 316
- Scope (V variables) ... 318
- Variable display ... 136

Velocity feedforward G918 ... 307
View of contour (simulation) ... 374

W

Wait for moment G204 ... 305
Warnings (simulation) ... 372
WHILE.. Program repeat ... 323
Width (of tool) ... 622
WINDOW (special output window) ... 312
Window selection
- Editing window (DIN PLUS) ... 113
- Setting the window (TURN PLUS) ... 400
- Simulation ... 369
- Window change (TURN PLUS) ... 392

WINDOWA (special output window) ... 313
WINDOWS networks ... 661
Working plan generation (TURN PLUS)
- AWG ... 534
- IWG ... 494

Working window ... 46
Workpiece blank contour
- DIN PLUS
  - Fundamentals ... 118
  - Workpiece-blank definition ... 146
- TURN PLUS
  - Contour elements ... 402
  - Editing the contours of a blank part ... 461
  - Input of ... 396

Workpiece datum
- Entering ... 76
- Fundamentals ... 40

Workpiece group G99 ... 284
Workpiece transfer
- C-angle offset G905 ... 287
- Controlled parting using lag error monitoring G917 ... 291
- Controlled parting using spindle monitoring G991 ... 292
- Measuring angular offset during spindle synchronization G906 ... 288
- Spindle synchronization G720 ... 286
- Traversing to a fixed stop G916 ... 288
- Values for controlled parting G992 ... 293

X

XY plane G17 (front or rear face) ... 723
XZ plane G18 (turning) ... 723

Y

Y axis - Basics ... 32

Z

Zero point
- Absolute shift G59 ... 201
- Additive shift G56 ... 200
- Changing in TURN PLUS ... 452
- Machine datum ... 40
- Parameter-dependent shift G53... G55 ... 199
- Relative shift G51 ... 199
- Shift in the simulation ... 366
- Shift in variables G902 ... 306
- Shift, activating G980 ... 309
- Shift, C axis G152 ... 254
- Shift, deactivating G920 ... 308
- Shift, deactivating tool lengths G921 ... 308
- Shifts and tool lengths, activating G981 ... 310
- Shifts, overview ... 198
- Workpiece datum ... 40

Zoom function
- Automatic mode (graphical display) ... 94
- Simulation ... 371
- TURN PLUS control graphics ... 547

Section Codes

Program section codes			Program section codes	
Program head			**Workpiece machining**	
PROGRAMMKOPF [PROGRAM HEAD]	Page 136		BEARBEITUNG [MACHINING]	Page 144
REVOLVER [TURRET]	Page 137		ZUORDNUNG [ASSIGNMENT]	Page 144
SCHEIBENMAGAZIN [PLATE MAGZN.]			ENDE [END]	Page 144
SPANNMITTEL [CHUCKING EQUIPMENT]	Page 142		**Subroutines**	
Contour definition			UNTERPROGRAMM [SUBPROGRAM]	Page 145
KONTUR [CONTOUR]	Page 143		RETURN	Page 145
ROHTEIL [WORKPIECE BLANK]	Page 143		**Others**	
FERTIGTEIL [FINISHED PART]	Page 143		CONST	Page 145
HILFSKONTUR [AUXILIARY CONTOUR]	Page 144			
C-axis contours			**Y-axis contours**	
STIRN [FRONT]	Page 144		STIRN_Y [FRONT_Y]	
RUECKSEITE [REAR SIDE]	Page 144		RUECKSEITE_Y [REAR SIDE_Y]	
MANTEL [SURFACE]	Page 144		MANTEL_Y [SURFACE_Y]	

Overview of G Commands in the CONTOUR Section

G commands for turning contours

Turning contour				Turning contour		
Workpiece-blank definition				**Contour form elements**		
G20-Geo	Chuck part, cylinder/tube	Page 146		G34-Geo	Thread (standard)	Page 159
G21-Geo	Cast part	Page 146		G37-Geo	Thread (general)	Page 160
Basic Contour Elements				G49-Geo	Bore hole at turning center	Page 162
G0-Geo	Starting point of contour	Page 147		**Help commands for contour definition**		
G1-Geo	Line segment	Page 147		Overview:Help commands for contour definition		Page 163
G2-Geo	Circular arc with incremental center dimensioning	Page 148		G7-Geo	Precision stop ON	Page 164
G3-Geo	Circular arc with incremental center dimensioning	Page 148		G8-Geo	Precision stop OFF	Page 164
G12-Geo	Circular arc with absolute center dimensioning	Page 150		G9-Geo	Precision stop blockwise	Page 164
G13-Geo	Circular arc with absolute center dimensioning	Page 150		G10-Geo	Peak-to-valley height	Page 164
Contour form elements				G38-Geo	Feed rate reduction factor	Page 165
G22-Geo	Recess (standard)	Page 152		G39-Geo	Attributes of overlay elements	Page 165
G23-Geo	Recess/relief turn	Page 153		G52-Geo	Blockwise oversize	Page 166
G24-Geo	Thread with undercut	Page 155		G95-Geo	Feed per revolution	Page 166
G25-Geo	Undercut contour	Page 156		G149-Geo	Additive compensation	Page 167

G commands for C-axis contours

C-axis contour				C-axis contour		
Overlapping contours				**Lateral surface contours**		
G308-Geo	Beginning of pocket/island	Page 168		G110-Geo	Starting point of lateral surface contour	Page 179
G309-Geo	End of pocket/island	Page 168		G111-Geo	Line segment, lateral surface	Page 179
Front and rear face contours				G112-Geo	Circular arc, lateral surface	Page 180
G100-Geo	Starting point of contour, face	Page 172		G113-Geo	Circular arc, lateral surface	Page 180
G101-Geo	Line segment, face	Page 172		G310-Geo	Bore hole, lateral surface	Page 181
G102-Geo	Circular arc on face	Page 173		G311-Geo	Linear slot, lateral surface	Page 182
G103-Geo	Circular arc on face	Page 173		G312-Geo	Circular slot on lateral surface	Page 182
G300-Geo	Bore hole, face	Page 174		G313-Geo	Circular slot on lateral surface	Page 182
G301-Geo	Linear slot, face	Page 175		G314-Geo	Full circle, lateral surface	Page 183

C-axis contour				C-axis contour		
G302-Geo	Circular slot on face	Page 175		G315-Geo	Rectangle, lateral surface	Page 183
G303-Geo	Circular slot on face	Page 175		G317-Geo	Eccentric polygon, lateral surface	Page 184
G304-Geo	Full circle, face	Page 176		G411-Geo	Linear pattern, lateral surface	Page 185
G305-Geo	Rectangle on face	Page 176		G412-Geo	Circular pattern, lateral surface	Page 186
G307-Geo	Eccentric polygon on face	Page 177				
G401-Geo	Linear pattern, face	Page 177				
G402-Geo	Circular pattern, face	Page 178				

G commands for Y-axis contours

Y-axis contour			Y-axis contour	
XY plane			YZ plane	
G170-Geo	Starting point of contour		G180-Geo	Starting point of contour
G171-Geo	Line segment		G181-Geo	Line segment
G172-Geo	Circular arc		G182-Geo	Circular arc
G173-Geo	Circular arc		G183-Geo	Circular arc
G370-Geo	Hole		G380-Geo	Hole
G371-Geo	Linear slot		G381-Geo	Linear slot
G372-Geo	Circular slot		G382-Geo	Circular slot
G373-Geo	Circular slot		G383-Geo	Circular slot
G374-Geo	Full circle		G384-Geo	Full circle
G375-Geo	Rectangle		G385-Geo	Rectangle
G377-Geo	Eccentric polygon		G387-Geo	Eccentric polygon
G471-Geo	Linear pattern		G481-Geo	Linear pattern
G472-Geo	Circular pattern		G482-Geo	Circular pattern
G376-Geo	Single surface		G386-Geo	Single surface
G477-Geo	Centric polygon		G487-Geo	Centric polygon

Overview of G Commands in the MACHINING Section

G commands for turning

Turning—Basic functions				Turning—Basic functions		
Tool positioning without machining				**Zero point shifts**		
G0	Positioning in rapid traverse	Page 187		G53	Parameter-dependent zero point shift	Page 199
G14	Move to the tool change position	Page 187		G54	Parameter-dependent zero point shift	Page 199
G701	Rapid traverse to machine coordinates	Page 187		G55	Parameter-dependent zero point shift	Page 199
Simple linear and circular movements				G56	Additive zero point shift	Page 200
G1	Linear movement	Page 189		G59	Absolute zero point shift	Page 201
G2	Circular arc with incremental center dimensioning	Page 190		G121	Contour mirroring/shifting	Page 202
G3	Circular arc with incremental center dimensioning	Page 190		G152	Zero point shift, C axis	Page 254
G12	Circular arc with absolute center dimensioning	Page 191		G920	Deactivate zero point shift	Page 308
G13	Circular arc with absolute center dimensioning	Page 191		G921	Deactivate zero point shift, tool dimensions	Page 308
Feed rate and spindle speed				G980	Activate zero point shift	Page 309
Gx26	Speed limit *	Page 192		G981	Activate zero point shift, tool dimensions	Page 310
G48	Acceleration (slope)	Page 192		**Oversizes**		
G64	Interrupted feed	Page 193		G50	Switch off oversize	Page 204
G192	Feed per minute for rotary axis	Page 193		G52	Switch off oversize	Page 204
Gx93	Feed per tooth *	Page 194		G57	Paraxial oversize	Page 204
G94	Feed per minute	Page 194		G58	Contour-parallel oversize	Page 205
Gx95	Feed per revolution	Page 194		**Safety clearances**		
Gx96	Constant surface speed	Page 195		G47	Set safety clearances	Page 206
Gx97	Shaft speed	Page 195		G147	Safety clearance (milling)	Page 206
G922	Shaft speed with V constant	Page 311				

Turning—Basic functions		
Tool-tip radius compensation (TRC/MCRC)		
G40	Switch off TRC/MCRC	Page 197
G41	TRC/MCRC, left	Page 197
G42	TRC/MCRC, right	Page 197
Zero point shifts		
Overview: Zero point shifts		Page 198
G51	Relative zero point shift	Page 199

Turning—Basic functions		
Tools, types of compensation		
T	Insert the tool	Page 207
G148	(Changing the) cutter compensation	Page 208
G149	Additive compensation	Page 209
G150	Compensate right tool tip	Page 210
G151	Compensate left tool tip	Page 210
G710	Adding tool dimensions	Page 211

Cycles for turning

Turning—Cycles		
Simple turning cycles		
G80	End of cycle	Page 231
G81	Simple longitudinal roughing	Page 231
G82	Simple face roughing	Page 232
G83	Contour repeat cycle	Page 234
G85	Undercut	Page 235
G86	Simple recessing cycle	Page 236
G87	Transition radii	Page 238
G88	Chamfer	Page 238
Drilling cycles		
G36	Tapping	Page 250
G71	Simple drilling cycle	Page 246
G72	Boring, countersinking, etc.	Page 248
G73	Tapping cycle	Page 249
G74	Deep-hole drilling cycle	Page 251

Turning—Cycles		
Contour-based turning cycles		
G810	Longitudinal roughing cycle	Page 212
G820	Face roughing cycle	Page 215
G830	Contour-parallel roughing cycle	Page 218
G835	Contour-parallel with neutral tool	Page 220
G860	Universal recessing cycle	Page 222
G866	Simple recessing cycle	Page 224
G869	Recess turning cycle	Page 225
G890	Finishing cycle	Page 228
Thread cycles		
G31	Thread cycle	Page 240
G32	Single thread cycle	Page 242
G33	Single thread cut (Thread single path)	Page 244
G933	Thread switch	Page 239
G799	Thread milling, axial	Page 278
G800	Thread milling in XY plane	
G806	Thread milling in YZ plane	

Synchronization commands

Synchronization				Synchronization		
Assigning the contour to the operation				**Spindle synchronization, workpiece transfer**		
G98	Assignment spindle – workpiece	Page 283		G30	Converting and mirroring	Page 282
G99	Workpiece group	Page 284		G121	Contour mirroring/shifting	Page 202
Slide synchronization				G720	Spindle synchronization	Page 286
G62	One-sided synchronization	Page 284		G905	Measuring C-angle offset	Page 287
G63	Synchronous start of slides	Page 285		G906	Measuring angular offset during spindle synchronization	Page 288
G162	Set synchronizing mark	Page 285		G916	Traversing to a fixed stop	Page 288
Contour follow-up				G917	Controlled parting using following error (lag error) monitoring	Page 291
G702	Storing/loading contour follow-up	Page 294		G991	Controlled parting using spindle monitoring	Page 292
G703	Contour follow-up ON/OFF	Page 294		G992	Values for controlled parting	Page 293
G706	K default branch	Page 295				

C-axis machining

C-axis machining				C-axis machining		
C axis				**Milling cycles**		
				G799	Thread milling, axial	Page 278
G119	Select C axis (No. of C axis)	Page 253		G801	Engraving, face	Page 279
G120	Reference diameter, lateral-surface machining	Page 253		G802	Engraving, lateral surface	Page 280
G152	Zero point shift, C axis	Page 254		G840	Contour milling	Page 261
G153	Standardize C axis	Page 254		G845	Pocket milling, roughing	Page 270
				G846	Pocket milling, finishing	Page 276
Face/rear-side machining				**Lateral-surface machining**		
G100	Rapid traverse, face	Page 255		G110	Rapid traverse, lateral surface	Page 258
G101	Linear path, face	Page 256		G111	Linear path, lateral surface	Page 259
G102	Circular arc, face	Page 257		G112	Circular arc, lateral surface	Page 260
G103	Circular arc, face	Page 257		G113	Circular arc, lateral surface	Page 260

Variable programming, program branches

Variable programming, program branches				Variable programming, program branches		
Programming with variables				**Data input and data output**		
# variables	Evaluation during program conversion	Page 316		INPUT	Input (# variables)	Page 312
V variable	Evaluation during program execution	Page 318		WINDOW	Open output window (# variables)	Page 312
Program branches, program repeats				PRINT	Output (# variables)	Page 313
IF..THEN..	Program branching	Page 322		INPUTA	Input (V variables)	Page 314
WHILE..	Program repeat	Page 323		WINDOWA	Open output window (V variables)	Page 313
SWITCH..	Program branching	Page 324		PRINTA	Output (V variables)	Page 314
Special functions				**Subroutines**		
$	Slide code	Page 326		Subprogram call		Page 327
/	Skip level	Page 326				

Measuring functions, load monitoring

Measuring functions, load monitoring				Measuring functions, load monitoring		
In-process measuring				**Post-process measuring**		
G910	Activate in-process measuring	Page 296		G915	Post-process measuring	Page 298
G912	Actual-value determination for in-process measuring	Page 297		**Load monitoring**		
G913	Deactivate in-process measuring	Page 297		G995	Define monitoring zone	Page 301
G914	Deactivate probe monitoring	Page 297		G996	Type of load monitoring	Page 301

Other G Functions

Other G Functions				Other G Functions		
G4	Dwell time	Page 302		G907	Block speed monitoring off	Page 306
G7	Precision stop ON	Page 302		G908	Feed rate override 100%	Page 307
G8	Precision stop OFF	Page 302		G909	Interpreter stop	Page 307
G9	Precision stop (blockwise)	Page 302		G918	Feedforward on/off	Page 307
G15	Move rotary axes	Page 303		G919	Spindle override 100%	Page 307
G60	Deactivate protection zone	Page 303		G920	Deactivate zero point shift	Page 308
G65	Display chucking equipment	Page 304		G921	Deactivate zero point shift, tool dimensions	Page 308
G66	Component position	Page 305		G930	Sleeve monitoring	Page 310
G204	Waiting for time	Page 305		G975	Lag error limit	Page 309
G717	Update nominal values	Page 305		G980	Activate zero point shift	Page 309
G718	Eliminate lag error	Page 306		G981	Activate zero point shift, tool dimensions	Page 310
G901	Actual values in variables	Page 306		G940	T no. internal	Page 308
G902	Zero point shift in variables	Page 306		G941	Transfer pocket compensation values	Page 309
G903	Lag error in variables	Page 306				

B-axis and Y-axis machining

Y-axis machining			Y-axis machining	
Working planes			**Milling cycles**	
G16	Tilting the working plane		G841	Area milling - roughing
G17	XY plane (front or rear face)		G842	Area milling - finishing
G18	XZ plane (turning)		G843	Centric polygon - roughing
G19	YZ plane (side view/lateral surface)		G844	Centric polygon - finishing
Tool positioning without machining			G845	Pocket milling, roughing
G0	Positioning in rapid traverse		G846	Pocket milling, finishing
G14	Move to the tool change position		G800	Thread milling in XY plane
G701	Rapid traverse to machine coordinates		G806	Thread milling in YZ plane
G714	Insert magazine tool		G803	Engraving in XY plane
G712	Define tool position		G804	Engraving in YZ plane
G600	Tool pre-selection		G808	Hobbing
Simple linear and circular movements				
G1	Linear path			
G2	Circular arc with incremental center dimensioning			
G3	Circular arc with incremental center dimensioning			
G12	Circular arc with absolute center dimensioning			
G13	Circular arc with absolute center dimensioning			

Overview of G Commands in the MACHINING Section

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☎ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
TNC support ☎ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☎ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☎ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
Lathe controls ☎ +49 8669 31-3105
E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

635 787-22 · Ver02 · pdf · 4/2010